CNC 8055 T

Operating Manual

Ref. 0001 (in)

Please note that some of the features described in this manual might not be implemented in the software version that you just obtained.

Probing canned cycles	Option
Tool life monitoring	Option
DNC	Option
Software for 4 axes	Option
Software for 7 axes	Option
Profile editor	Option
"C" axis (Lathe)	Option
Tangential Control	Option
Conversational software (TC and TCO)	Option

---------- o ----------

The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

INDEX

VERSION HISTOTY

INTRODUCTION

1. OVERVIEW

1.1 Part-programs 1
1.2 Monitor information layout 4
1.3 Keyboard layout 6
1.4 Operator panel layout 8

2. OPERATING MODES

2.1 Help systems 3
2.2 Software Update 5

3. EXECUTE / SIMULATE

3.1 Block selection and stop condition 4
3.2 Display selection 6
3.2.1 Standard display mode 7
3.2.2 Position display mode 8
3.2.3. Part program display mode 9
3.2.4. Subroutine display mode 9
3.2.5 Following error display mode 12
3.2.6 User display mode 12
3.2.7 Execution time display mode 13
3.3 MDI 15
3.4 Tool inspection 16
3.5 Graphics 18
3.5.1 Type of graphics 19
3.5.2 Display area 22
3.5.3 Zoom 23
3.5.4 Graphic parameters 24
3.5.5 Clear screen 26
3.5.6 Deactivate graphics 26
3.5.7 Measure 27
3.6 Single block 28

4. EDIT

4.1 Edit 2
4.1.1 Editing in CNC language 2
4.1.2 Teach-in editing 3
4.1.3 Interactive editor 4
4.1.4 Profile editor 5
4.1.4.1 Operation with the profile editor 6
4.1.4.2 Profile editing 7
4.1.4.3 Definition of a straight section 8
4.1.4.4 Definition of a circular section 9
4.1.4.5 Corners 10
4.1.4.6 Modify 11
4.1.4.7 Finish 13
4.1.4.8 Examples of profile definition 14
4.2 Modify 16
4.3 Find 17
4.4 Replace 18
4.5 Delete block 19
4.6 Move block 20
4.7 Copy block 21
4.8 Copy a program 22
4.9 Include a program 23
4.10 Editor parameters 24
4.10.1 Autonumbering 24
4.10.2 Axes selection for teach-in editing 25

5. JOG

5.1 Jogging the axes 10
5.1.1 Continuous jog 10
5.1.2 Incremental jog 11
5.1.3 Jogging with electronic handwheel 12
5.1.3.1 General handwheel 14
5.1.3.2 Individual handwheel 14
5.1.3.3 PATH handwheel 15
5.2 Manual control of the spindle 16

6. TABLES

6.1 Zero offset table 2
6.2 Tool magazine table 3
6.3 Tool table 4
6.4 Tool offset table 6
6.5 Tool geometry table 10
6.5.1 How to edit the tool geometry table 13
6.6 Global and local parameter tables 14
6.7 How to edit tables 15

7. UTILITIES

7.1 Directory 1
7.1.1 Directory of the external devices 3
7.2 Copy 4
7.3 Delete 4
7.4 Rename 5
7.5 Protections 6
7.6 Change date 7

8. DNC

8.1 CNC 1
8.2 DNC 2

9. PLC

9.1 Edit 2
9.2 Compile 9
9.3 Monitoring 10
9.3.1 Monitoring with the PLC in operation and with the PLC stopped 17
9.4 Active messages 19
9.5 Active pages (screens) 19
9.6 Save program 19
9.7 Restore program 20
9.8 Resources in use 20
9.9 Statistics 21
9.10 Logic analyzer 23
9.10.1 Description of the work screen 23
9.10.2 Selection of variables and trigger conditions 26
9.10.2.1 Variable selection 26
9.10.2.2 Selection of trigger condition 28
9.10.2.3 Selection of time base 30
9.10.3 Execute trace 31
9.10.3.1 Data capture 32
9.10.3.2 Modes of operation 33
9.10.3.3 Trace representation 34
9.10.4 Analyze trace 35

10. SCREEN EDITOR

10.1 Utilities 3
10.2 Editing custom screens (pages) and symbols 5
10.3 Graphic elements 10
10.4 Texts 15
10.5 Modifications 18

11. MACHINE PARAMETERS

11.1 Machine parameter tables 2
11.2 Miscellaneous function tables 3
11.3 Leadscrew error compensation tables 4
11.4 Cross compensation tables 5
11.5 Operation with parameter tables 6

12. DIAGNOSIS

12.1 Configuration 2
12.1.1 Hardware configuration 2
12.1.2 Software configuration 3
12.2 Hardware test 4
12.3 Memory test 5
12.4 Flash memory test 5
12.5 User 5
12.6 Hard disk 5
12.7 Interesting Notes 6

VERSION HISTORY (T)

(LATHE MODEL)

Date: **December 1999** ***Software Version:*** **4.0x**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Portuguese language	Installation manual	Chap. 3
Tangential control	Installation manual Programming manual	Chap. 9, Chap. 10, Appendix Chap. 6, Chap. 11, Appendix
PLC. user registers from R1 to R499	Installation manual Programming manual	Chap. 6, Chap. 7, Appendix Chap. 11
CNC status screen	Operating manual	Chap. 8
Hard Disk (HD)	Installation manual	Chap. 1, Chap. 3, Appendix
Diagnosis of the HD	Operating manual	Chap. 12,
Integrate the HD in an external PC network	Installation manual	Chap. 3
Consult directories, delete, rename and copy programs in the same or another device.	Operating manual Programming manual	Chap. 1, Chap. 7 Chap. 1
Execution and simulation from RAM, Memkey Card, HD or serial line.	Operating manual	Chap. 1, Chap. 3,
It is now possible to execute (EXEC) and open (OPEN) for editing a program stored in any device.	Programming manual	Chap. 14, Appendix
Thread repair. Reference (home) the spindle before.	Programming manual TC operating manual	Chap. 9 Chap. 4
Simulation in rapid, without assuming G95 or M3, M54, etc.	Operating manual	Chap. 3
Geometry associated with the tool offset.	Installation manual Operating manual	Chap. 3 Chap. 6
Live tool with M45 or as if it were a 2nd spindle	Installation manual	Chap. 3
PLC channel affected by another feedrate override set by PLC.	Installation manual	Chap. 11
Independent x1, x10, x100 factor for each handwheel.	Installation manual Programming manual	Chap. 4, Chap. 10, Appendix Chap. 11
Handling the Fagor HBE handwheel	Installation manual	Chap. 4, 9, 10, Appendix
Spindle synchronization (G77 S)	Installation manual Programming manual	Chap. 3, 9, 10, Appendix Chap. 5, 11, Appendix
Optimizing of profile machining.	Programming manual TC operating manual	Chap. 9 Chap. 4
(2) axes controlled by a single servo drive	Installation manual	Chap. 3, 4, 9, Appendix
G75 function affected by Feedrate override (%)	Installation manual	Chap. 3
Probe. Probe position by cycle parameters.	Programming manual	Chap. 10
Protection against deleting OEM screens	Operating manual	Chap. 7
Detecting temperature and battery voltage on the new CPU.		
TC option. ISO program management, also like MDI.	TC operating manual	Chap. 3
TC option. Coolant icon in all cycles.	TC operating manual	Chap. 4
TC option. Background editing.	TC operating manual	Chap. 4
TC option. Key codes for user cycles.	TC operating manual	Appendix

INTRODUCTION

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Protections of the unit itself

Power Supply Module

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input

Axes module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against reverse connection of the power supply.

Input / Output Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Input / Output and Tracing Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Fan Module

It carries 1 or 2 external fuses depending on model
The fuses are fast (F), of 0.4 Amp./ 250V. to protect the fans.

Monitor

The type of protection fuse depends on the type of monitor. See the identification label of the unit itself.

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

Safety symbols

Symbols which may appear on the manual

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

"Electrical Shock" symbol
It indicates that point may be under electrical voltage

"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the Monitor or the Central Unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

When sending the monitor, especially protect the CRT glass

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE CNC

OEM Manual	Is directed to the machine builder or person in charge of installing and starting-up the CNC.
USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual describing how to operate the CNC. Programming Manual describing how to program the CNC.
DNC Software Manual	Is directed to people using the optional DNC communications software.
DNC Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the CNC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The Operating Manual for the Lathe model CNC contains the following chapters:

Index

New Features and Modifications for the Lathe Model

Introduction
Summary of safety conditions
Shipping terms
Fagor documentation for the CNC.
Manual contents

Chapter 1 Overview
Location of part-programs, how to edit and execute them.
Layout of the keyboard, operator panel and of the information on the monitor.

Chapter 2 Operating modes
Description of the different operating modes of the CNC.

Chapter 3 Execute/ Simulate
It describes how to operate in the “Execution” and “Simulation” modes.
Both operations may be performed in automatic or single block mode.

Chapter 4 Edit
Description of the “Edit” mode of operation.
The different ways to edit a part-program are: in CNC language, in Teach-in mode, using the Interactive editor and the Profile editor.

Chapter 5 JOG
Description of the “JOG” mode of operation..
This is the operating mode to be used whenever the machine is to be controlled manually to move the axes of the machine as well as to control the spindle.

Chapter 6 Tables
Description of the “Tables” mode of operation.
It allows access to the various data tables of the CNC: Zero offsets, Tool offsets, Tool table, tool magazine, and global and local arithmetic parameters.

Chapter 7 Utilities
Description of the “Utilities” mode of operation”.
It allows access to the directory of part-programs, subroutines and to the part-program directory of the PC or peripheral device connected to the CNC. It is also possible to copy, delete, move or rename part-programs.
It indicates the protections that could be assigned to a part-program.
It shows the various ways to operate with the EEPROM. memory.

Chapter 8 Status
It shows the status of the CNC and the DNC communication lines.
Description of the “DNC” mode of operation and how to operate via the serial lines

Chapter 9 PLC
Description of the “PLC” mode of operation.
It indicates how to edit and compile the PLC program.
It is possible to verify how the PLC program works and the status of its numerous variables.
It shows the date the PLC program was edited, its memory size and the execution times (cycle times) for its different modules.
It offers a detailed description of the logic analyzer.

Chapter 10 Graphic Editor

Description of the “Graphic Editor” mode of operation”.
It indicates how to create user defined pages (screens) and symbols to create user screens.
It shows how to use user pages in customizing programs, how to display a user page on power-up and how to activate user pages from the PLC.

Chapter 11 Machine Parameters

Description of the “Machine Parameters “ mode of operation”.
It is possible to access and operate with the tables for: Machine parameters, miscellaneous “M” functions, leadscrew error compensation and cross compensation.

Chapter 12 Diagnosis

Description of the “Diagnosis” mode of operation”.
It is possible to know the CNC configuration and run a system test.

1. OVERVIEW

In this manual an explanation is given of how to operate the CNC by means of its Monitor-Keyboard unit and the Operator Panel.

The Monitor-Keyboard unit consists of:

* The Monitor or CRT screen, which is used to show the required system information.

* The Keyboard, which allows communication with the CNC, allowing information to be requested by means of commands or by changing the CNC status by generating new instructions.

1.1 PART-PROGRAMS

Editing

To create a part-program, access the Edit mode. See chapter 5 in this manual.

The new part-program edited is stored in the CNC's RAM memory.

A copy of the part-programs may be stored in the "MemKey Card", at a PC connected through serial line 1 or 2 or in the hard disk (HD module). See chapter 7 in this manual.

When using a PC through serial line 1 or 2, proceed as follows:
- Execute the "Fagor50.exe" applications program at the PC.
- Activate DNC communications at the CNC. See chapter 8 in this manual.
- Select the work directory as shown in chapter 7 of this manual. Option: Utilities\ Directory\ Serial L.\ Change directory.

With the Edit mode of operation, part-programs residing in the CNC's RAM memory may be modified. To modify a program stored in the "MemKey Card", in a PC or in the hard disk, it must be previously copied into RAM memory.

Execution

Part-programs stored anywhere may be executed or simulated. See chapter 3 in this manual.

The user customizing programs must be in RAM memory so the CNC can execute them.

The GOTO and RPT instructions cannot be used in programs executed from a PC connected through the serial lines. See chapter 14 of the programming manual.

The subroutines can only be executed if they reside in the CNC's RAM memory. Therefore, to execute a subroutine stored in the "MemKey Card", in a PC or in the hard disk, it must be first copied into the CNC's RAM memory.

From a program in execution, another program can be executed which is in RAM memory, in the "MemKey Card", in a PC or in the hard disk using the EXEC instruction. See chapter 14 of the programming manual.

Utilities

This operating mode, chapter 7 of this manual, lets display the part-program directory of all the devices, make copies, delete, rename and even set the protections for any of them.

Ethernet

When having the Ethernet option and if the CNC is configured as another node within the computer network, the following operations are possible from any PC of the network:

- Access the part-program directory of the Hard Disk(HD).
- Edit, modify, delete, rename, etc.the programs stored on the hard disk (HD).
- Copy programs from the hard disk to the PC and vice versa.

To configure the CNC as another node within the computer network, see section 3.3.4 of the installation manual.

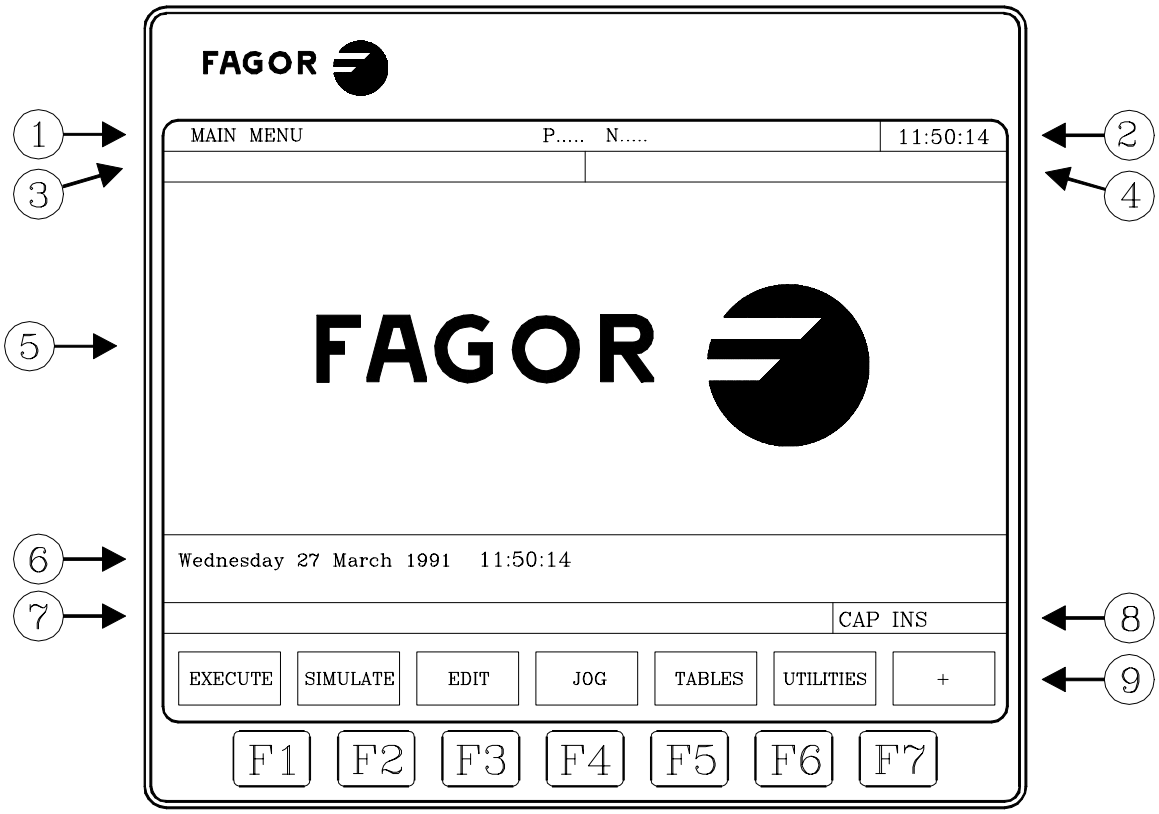
Operations that may be carried out with part-programs:

	RAM Memory	CARD A	HD	DNC
Consult the program directory in ...	Yes	Yes	Yes	Yes
Consult the subroutine directory in ...	Yes	No	No	No
Create work directory in ..	No	No	No	No
Change work directory in ..	No	No	No	Yes
Edit a program in ..	Yes	No	No	No
Modify a program in ..	Yes	No	No	No
Delete a program from ..	Yes	Yes	Yes	Yes
Copy from/to RAM memory to/from ...	Yes	Yes	Yes	Yes
Copy from/to CARD A to/from ...	Yes	Yes	Yes	Yes
Copy from/to HD to/from ...	Yes	Yes	Yes	Yes
Copy from/to DNC to/from ...	Yes	Yes	Yes	Yes
Rename a program in ..	Yes	Yes	Yes	No
Change the comment of a program in ..	Yes	Yes	Yes	No
Change protections of a program in ..	Yes	Yes	Yes	No
Execute a part-program in ..	Yes	Yes	Yes	Yes
Execute a user program in ..	Yes	No	No	No
Execute the PLC program in ..	Yes	*	No	No
Execute programs using the GOTO or RPT instructions from ..	Yes	Yes	Yes	No
Execute subroutines stored in ..	Yes	No	No	No
Execute programs stored in RAM, CARD A or HD using the EXEC instruction from ..	Yes	Yes	Yes	Yes
Execute programs via DNC with the EXEC instruction from ..	Yes	Yes	Yes	No
Open programs stored in RAM, CARD A or HD using the OPEN instruction from ..	Yes	Yes	Yes	Yes
Open programs via DNC using the OPEN instruction from ..	Yes	Yes	Yes	No
Consult from a PC and through Ethernet, the program directory in ...	No	No	Yes	No
Consult from a PC and through Ethernet, the subroutine directory in ...	No	No	No	No
Create from a PC and through Ethernet, a directory in...	No	No	No	No

* If it is not in RAM memory, it generates an executable code in RAM and it executes it..

1.2 MONITOR INFORMATION LAYOUT

The monitor is divided into the following areas or display windows:



1.- This window indicates the selected operating mode, as well as the program number and the number of active blocks.

The program status is also indicated (in execution or interrupted) and if the DNC is active.

2.- This window indicates the time in the " hours : minutes : seconds " format.

3.- This window displays the Messages sent to the operator from the part program or via DNC.

The last message received will be shown regardless of where it has come from.

4.- This window will display messages from the PLC.

If the PLC activates two or more messages, the CNC will always display the one with the highest priority, which is the message with the smallest number. In this way, MSG1 will have the highest priority and MSG128 will have the lowest.

In this case the CNC will display the character + (plus sign), indicating that there are more messages activated by the PLC, it being possible to display them if the ACTIVE

MESSAGE option is accessed in the PLC mode.

In this window the CNC will also display the character ***** (asterisk), to indicate that at least one of the 256 user-defined screens is active.

The screens which are active will be displayed, one by one, if the ACTIVE PAGES option is accessed in the PLC mode.

5.- Main window.

Depending on the operating mode, the CNC will show in this window all the information necessary.

When a CNC or PLC error is produced the system displays this in a superimposed horizontal window.

The CNC will always display the most important error and it will show:

* The "down arrow" key to indicate that another less important error has also occurred and to press this key to view its message.

* The "up arrow" key to indicate that another more important error has also occurred and to press this key to view its message.

6.- Editing window.

In some operating modes the last four lines of the main window are used as editing area.

7.-CNC communications window (errors detected in edition, nonexistent program, etc.)

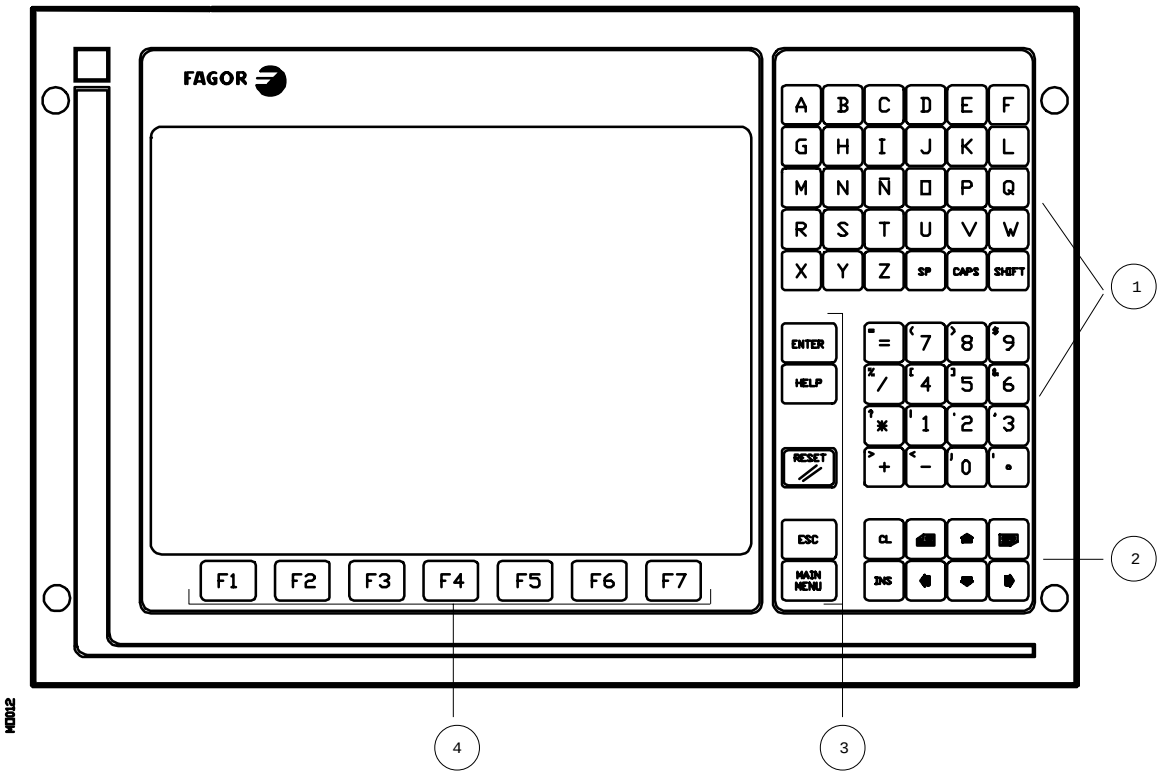
8.- This window displays the following information:

SHF	Indicates that the SHIFT key has been pressed to activate the second function of the keys. For example, if key is pressed after the SHIFT key, the CNC will understand that the "$" character is required.
CAP	This indicates capital letters (CAPS key). The CNC will understand that capital letters are required whenever this is active.
INS/REP	Indicates if it is insert mode (INS) or substitution (REP) mode. It is selected by means of the INS key.
MM/INCH	Indicates the unit system (millimeters or inches) selected for display.

9.- Shows the different options which can be selected with soft-keys F1 thru F7.

1.3 KEYBOARD LAYOUT

In accordance with the use of the different keys, it can be understood that the CNC keyboard is divided in the following way:



1.- Alphanumeric keyboard for the data entry in memory, selection of axes, tool offset, etc.

2.- Keys which allow the information shown on screen to be moved forward or backward, page to page or line to line, as well as moving the cursor all over the screen.

The CL key allows the character over which the cursor is positioned or the last one introduced, if the cursor is at the end of the line, to be erased.

The INS key allows the insert or substitution mode to be selected.

3.- Group of keys which due to their characteristics and importance are detailed below:

ENTER Used to validate CNC and PLC commands generated in the edition Window.

HELP Allows access to the help system in any operating mode.

RESET Used for initializing the history of the program in execution, by assigning it the values defined by machine parameters. It is necessary for the program to be stopped for the CNC to accept this key.

ESC Allows going back to the previous operating option shown on the monitor.

MAIN MENU When this key is pressed we can access the main CNC menu directly.

4.- SOFTKEYS or function keys which allow different operating options to be selected and which are shown on the monitor.

In addition, there are the following special keyboard sequences:

SHIFT RESET The result of this keystroke sequence is the same as if the CNC is turned off and turned back on. This option must be used after modifying the machine parameters of the CNC for these to be effective.

SHIFT CL With this keystroke sequence the display on the CRT screen disappears. To restore the normal state just press any key.

If, when the screen is off, an error is produced or a message from the PLC or CNC is received, the normal status of the screen will be restored.

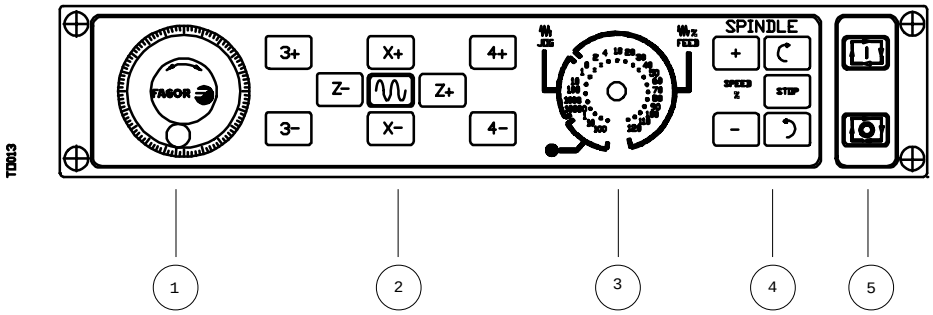
SHIFT This allows the position of the axes to be displayed on the right hand side of the screen as well as the status of the program being executed.

This can be used in any operating mode.

In order to recover the previous display it is necessary to press the keys using the same sequence.

1.4 OPERATOR PANEL LAYOUT

According to the utility which the different parts have, it can be considered that the Operator Panel of the CNC is divided in the following way:



1.- Position of the emergency button or electronic handwheel.

2.- Keyboard for manual movement of axes.

3.- Selector switch with the following functions:

Select the multiplication factor of the number of pulses from the electronic handwheel (1, 10 or 100).

Select the incremental value of the movement of the axes in movements made in the "JOG" mode.

Modify the programmed axis feedrate between 0% and 120%

4.- Keyboard which allows the spindle to be controlled, it being possible to activate it in the desired direction, stop it or vary the programmed turning speed between percentage values established by means of spindle machine parameters "MINSOVR" and "MAXOVR", with an incremental step established by means of the spindle machine parameter "SOVRSTEP".

5.- Keyboard for CYCLE START and CYCLE STOP of the block or program to be executed.

2. *OPERATING MODES*

After turning on the CNC, or after pressing the sequence of SHIFT-RESET keys, the FAGOR logo will appear in the main window of the monitor or the screen previously prepared as page 0 by means of the GRAPHIC EDITOR.

If the CNC shows the message **" Initialize? (ENTER / ESC) "**, it should be borne in mind that after pressing the ENTER key, all the information stored in memory and the machine parameters are initialized to default values indicated in the installation manual.

On the lower part of the screen the main CNC menu will be shown, it being possible to select the different operating modes by means of the softkeys F1 thru F7.

Whenever the CNC menu has more options than number of softkeys (7), the character "+" will appear in softkey f7. If this softkey is pressed the CNC will show the rest of the options available.

The options which the main CNC menu will show after turning it on, after pressing the key sequence SHIFT-RESET or after pressing the "MAIN MENU" softkey are:

EXECUTE Allows the execution of part programs in automatic or single block.

SIMULATE Allows simulation of parts programs in several modes.

EDIT Allows editing new and already-existing part programs.

JOG Allows manual control of the machine by means of the Control Panel keys.

TABLES Allows CNC tables relating to part programs (Zero Offsets, Tool Offsets, Tools, Tool Magazine and global or local arithmetic parameters) to be manipulated.

UTILITIES Allows program manipulation (copy, delete, rename, etc.)

STATUS It shows the CNC status and that of the DNC communication lines. It also lets activate and deactivate the communication with a PC through DNC.

DNC Allows communication with a computer via DNC to be activated or deactivated.

PLC Allows operation with the PLC (edit the program, monitor, change the status of its variables, access to the active messages, errors, pages, etc).

GRAPHIC EDITOR Allows, by means of a simple graphics editor, the creation of user-defined screens (pages), which can later be activated from the PLC, used in customized programs or presented when the unit is powered on (page 0).

MACHINE PARAMETERS Allows the machine parameters to be set to adapt the CNC to the machine.

DIAGNOSIS Makes a test of the CNC.

While the CNC is executing or simulating a part program it allows any other type of operating mode to be accessed without stopping the execution of the program.

In this way it is possible to edit a program while another is being executed or simulated.

It is not possible to edit the program which is being executed or simulated, nor execute or simulate two part programs at the same time.

2.1 HELP SYSTEMS

The CNC allows access to the help system (main menu, operating mode, editing of commands, etc.) at any time.

To do this, you must press the HELP key and the corresponding help page will be shown in the main window of the screen.

If the help consists of more than one page of information, the symbol indicating that this key can be pressed to access the following page or the indicating that it is possible to press this key to access the previous page.

The following help is available:

*** OPERATING HELP**

This is accessed from the operating mode menu, or when one of these has been selected but none of the options shown have been selected. In all these cases, the softkeys have a blue background color.

It offers information on the operating mode or corresponding option.

While this information is available on screen it is not possible to continue operating the CNC via the softkeys, it being necessary to press the HELP key again to recover the information which was on the main screen before requesting help and continuing with the operation of the CNC.

The help system can also be abandoned by pressing the ESC key or the MAIN MENU key.

*** EDITING HELP**

This is accessed once one of the editing options has been selected (part programs, PLC program, tables, machine parameters, etc.) In all these cases, the softkeys have a white background.

It offers information on the corresponding option.

While this information is available, it is possible to continue operating with the CNC.

If the HELP key is pressed again, the CNC analyzes if the present editing status corresponds to the same help page or not.

If another page corresponds to it, it displays this instead of the previous one and if the same one corresponds, it recovers the information which was in the main window before requesting help.

The help menu can also be abandoned after pressing the ESC key, to return to the previous operating option, or the MAIN MENU key to return to the main menu.

* CANNED CYCLES EDITING HELP

It is possible to access this help when editing a canned cycle.

It offers information on the corresponding canned cycle and an editing assistance for the selected canned cycle is obtained at this point.

For the user's own cycles a similar editing assistance can be obtained by means of a user program. This program must be prepared with screen customizing instructions.

Once all the fields or parameters of the canned cycle have been defined the CNC will show the information which exists in the main window before requesting help.

The canned cycle which is programmed by means of editing assistance will be shown in the editing window, and the operator can modify or complete this block before entering it in memory by pressing the ENTER key.

Editing assistance can be abandoned at any time by pressing the HELP key. The CNC will show the information which existed on the main window before requesting help and allows programming of the canned cycle to continue in the editing window.

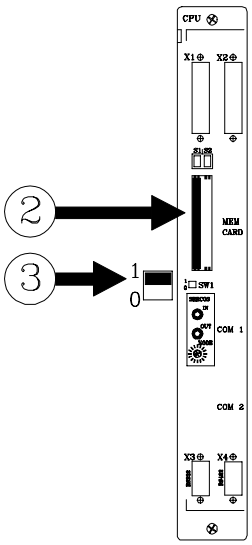
The help menu can also be abandoned after pressing the ESC key, to return to the previous operating option, or the MAIN MENU key to return to the main menu.

2.2 SOFTWARE UPDATE

Procedure

1- Turn the CNC off

2.- Replace the memory card in "Slot A" with the one containing the new software version.



3.- Set the SW1 switch to "1".

4- Turn the CNC on.

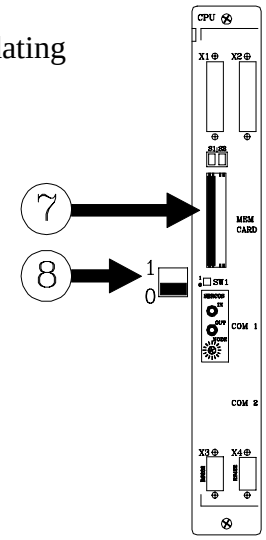
The screen will show the software updating page with the following information:

Installed version and New version
Checksum of the installed version and that of the new one.

5.- Press the [Update software] softkey

The CNC will display the various stages of the software updating process and their status.

When done with the updating process, the CNC will display a new screen with the steps to follow.



6.- Turn the CNC off

7.- Replace the memory card in "Slot A" with the "Memkey Card".

8.- Set the SW1 switch to “0”.

9- Turn the CNC on. The software version is now updated.

Notes:

With the memory card that contains the software version, the CNC CANNOT executed anything.

If the CNC is turned on with the "Memkey Card" in and the SW1 switch set to "1", the CNC does not come on, but its data is NOT affected.

Warning:

Reinstall the CNC software when replacing the Hard Disk module

The CNC software and the Hard Disc module must be compatible.

3. *EXECUTE / SIMULATE*

The EXECUTE operating mode allows the execution of part programs in automatic mode or in single block mode.

The SIMULATE operating mode allows the simulation of part-programs in automatic or single block mode.

When selecting one of these operating modes, one must indicate the location of the part-program to be executed or simulated.

The part program may be stored in the CNC's internal RAM memory, in the "Memkey Card", in PC connected through serial line 1 or 2, or in the hard disk (HD module).

After pressing one of these softkeys, the CNC displayes the corresponding part-program directory.

The program may be selected by:

- Keying in its number and pressing [ENTER] or
- Positioning the cursor of the scren over the desired program and pressing [ENTER].

When wished to SIMULATE a part-program, the CNC will request the type of simulation to be carried out as shown on the next page.

The executing or simulating conditions (fist block, type of graphics, etc.) may be set before executing or simulating the part-program. These conditions may also be modified if the execution or simulation is interrupted.

To execute or simulate a part-program, press

Note: To switch to JOG mode once executed or simulated a part program (or a section of it), the CNC will maintain the machining conditions (type of movement, feedrates, etc.) selected while executing or simulating it.

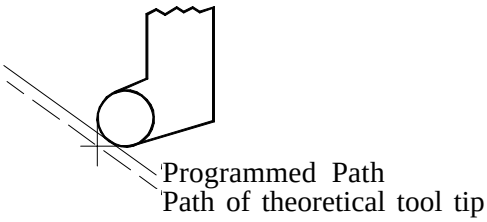
When simulating a part program, the CNC will ask for the type of simulation desired offering the following Options:

THEORETICAL PATH

- It ignores tool radius compensation (functions G41, G42) thus showing the graphic representation of the programmed path.
- It does not output the M, S, T function to the PLC.
- It does not move the machine axes or start the spindle.

G FUNCTIONS

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the theoretical tool nose (it does not coincide with the programmed path).



- It does not output the M, S, T functions to the PLC.
- It does not move the machine axes or start the spindle.

G, M, S, T FUNCTIONS

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the theoretical tool nose.
- It outputs the M, S, T functions to the PLC.
- It does not move the machine axes or start the spindle.

RAPID

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the theoretical tool nose.
- It outputs the M, S, T functions to the PLC.
- It starts the spindle if it has been programmed.
- The axes are moved at maximum feedrate allowed F0 regardless of the programmed F value and it can be varied using the Feedrate Override Switch.

RAPID [S=0]

- It takes into account tool radius compensation (functions G41, G42) thus showing the graphic representation of the path for the theoretical tool nose.
- It does not start the spindle.
- It does not output the M functions associated with the spindle when operating in open loop (rpm): M3, M4, M5, M41, M42, M43 and M44.
- It does output to the PLC the M function associated with spindle orientation (M19) when operating in closed loop.
- It outputs to the PLC the rest of the functions M, S, T.
- The axes, "C" axis included, are moved at maximum feedrate F0 regardless of the programmed F value and it can be varied using the Feedrate Override Switch.

	Graphics	Axis movement	Spindle control	Outputs M, S, T to the PLC	Outputs M3-4-5 M41-42-43-44
Theoretical Path	Programmed	No	No	No	No
G functions	Theoretical tip	No	No	No	No
G, M, S, T functions	Theoretical tip	No	No	Yes	Yes
Rapid	Theoretical tip	Yes	Yes	Yes	Yes
Rapid [S=0]	Theoretical tip	Yes	No	Yes	No

The executing or simulating conditions (initial block, type of graphics, etc.) that may be set before or while executing or simulating a part-program are:

BLOCK SELECTION

It allows selecting the block in which the execution or the simulation of the program will start.

STOP CONDITION

It allows selecting the block in which the execution or the simulation of the program will stop.

DISPLAY SELECTION

It allows the display mode to be selected.

MDI

It allows any type of block (ISO or high level) to be edited with programming assistance by means of softkeys.

Once a block has been edited and after pressing the key (cycle start), the CNC will execute this block without leaving this operating mode.

TOOL INSPECTION

Once the execution of the program has been interrupted, this option allows the tool to be inspected and changed should this be necessary.

GRAPHICS

This option carries out a graphic representation of the part during the execution or simulation of the selected part program.

It also allows selecting the type of graphic, the area to be displayed, the viewpoint and graphic parameters.

SINGLE BLOCK

Allows the part program to be executed one block at a time or continuously.

3.1 BLOCK SELECTION AND STOP CONDITION

The CNC will start to execute the required block from the first line of the program and will finish it when one of the program end functions M02 or M30 is executed.

If it is required to modify one of these conditions the **BLOCK SELECTION** and **STOP CONDITION** functions must be used.

BLOCK SELECTION

With this option it is possible to indicate the beginning block of the selected program execution or simulation. This cannot be used when the CNC is already executing or simulating the selected program.

When this option is selected, the CNC will show the selected program since the initial block must always belong to this program.

The operator must select with the cursor the block where the execution or simulation of the program will be started.

To do this, the cursor can be moved line by line with the up and down arrow keys or page by page with the page-up and page-down keys.

The “find” softkey options are also available:

BEGINNING: By pressing this key, the cursor will position at the first line of the program.

END: By pressing this key, the cursor will position at the last line of the program.

TEXT: With this function it is possible to search for a text or character sequence starting at the current cursor position.

When this softkey is pressed, the CNC requests the character sequence to be found.

Once this text has been keyed in, press the "**END OF TEXT**" softkey and the cursor will position over the first occurrence of the keyed text.

The found text will be highlighted and it will be possible to continue (by pressing "**ENTER**") with the search all along the program or quit by pressing either the "**ESC**" key or "**ABORT**" softkey.

The search can be done as many times as it is desired. Once searched to the end of the program, it will continue the search from the beginning.

When quitting the search mode, the cursor will be positioned at the last matching text found.

LINE NUMBER: After pressing this key, the CNC will request the number of the line to be found. Key in the desired line number and press **ENTER**. The cursor will, then, be positioned at the desired line.

Once the desired starting block has been selected, press **ENTER** to validate it.

STOP CONDITION

With this option it is possible to indicate the final execution or simulation block of the selected program. This cannot be used when the CNC is already executing or simulating the selected program.

When selecting this option, the CNC will show the following softkey functions:

PROGRAM SELECTION

This option will be used when the final execution or simulation block belongs to another program or to a subroutine resident in another program.

The CNC shows the part-program directory of the RAM memory. Use the cursor to select the desired program and press ENTER.

Then, carry out the BLOCK SELECTION as described next.

BLOCK SELECTION

Use the cursor to select the last program block to be executed.

Use the up and down arrow keys or page by page with the page-up and page-down keys.

The "find" softkey options are also available:

BEGINNING: By pressing this key, the cursor will position at the first line of the program.

END: By pressing this key, the cursor will position at the last line of the program.

LINE NUMBER: After pressing this key, the CNC will request the number of the line to be found. Key in the desired line number and press **ENTER**. The cursor will, then, be positioned at the desired line.

Once the desired starting block has been selected, press **ENTER** to validate it.

NUMBER OF TIMES

This function will be used to indicate that the execution or simulation of the selected program must stop after executing the "end block" a specific number of times.

When selecting this function, the CNC will request the number of times to be executed or simulated.

If a canned cycle or a call to a subroutine has been selected as the end block of the program, the CNC will stop after executing the complete canned cycle or the indicated subroutine.

If the selected block has a number of block repetitions, the program will stop after doing all the repetitions indicated.

3.2 DISPLAY SELECTION

With this option, it is possible to select the most appropriate display mode at any time even during execution or simulation of a part program.

The display modes available at the CNC and which can be selected with softkeys are:

STANDARD
POSITION
PART PROGRAM
SUBROUTINES
FOLLOWING ERRORS
USER
EXECUTION TIMES

All the display modes have a window at the bottom of the CRT which shows the history with the conditions in which machining is being done. The information shown is as follows:

F and **%** Programmed feedrate and selected feedrate OVERRIDE %.

S and **%** Programmed speed and OVERRIDE % of the selected spindle.

T Number of active tool.

D Number of active tool offset.

S RPM Real speed of the spindle in RPM.

When working in M19 this indicates the position of the spindle in degrees.

S M/MIN Real speed of the spindle in Meters/min. or feet/min.

G All active displayable G functions.

M All active M functions.

PARTC Parts counter. It indicates the number of consecutive parts executed with the same part-program.

Every time a new program is selected, this variable is reset to "0".

With this CNC variable (PARTC) it is possible to modify this counter from the PLC, from the CNC program and via DNC.

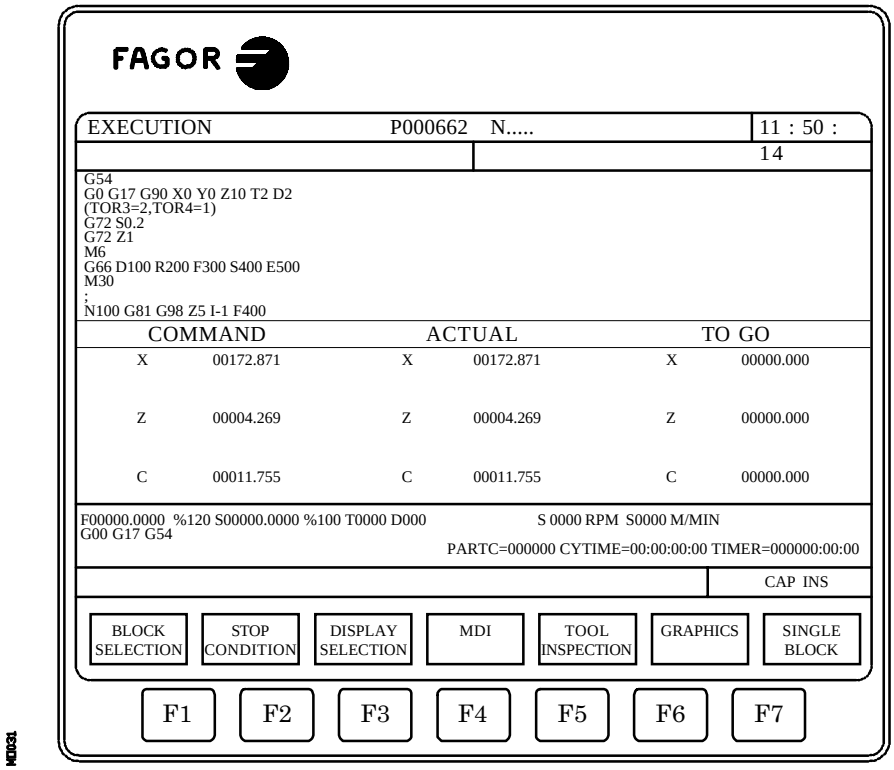
CYTIME Time elapsed during the execution of the part in “hours : minutes : seconds : hundredths of a second” format.

Every time a part-program execution starts, even when repetitive, this variable is reset to "0".

TIMER Time indicated by the PLC-enabled clock in “hours: minutes : seconds” format.

3.2.1 STANDARD DISPLAY MODE

This display mode is assumed by default on power-up and after the key sequence SHIFT-RESET and it shows the following fields or windows:



* A group of program blocks. The first of them is the block being executed.

* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and the format defined with the axis machine parameter "DFORMAT".

Each axis is provided with the following fields:

COMMAND. Indicates the programmed coordinate or position value which the axis must reach.

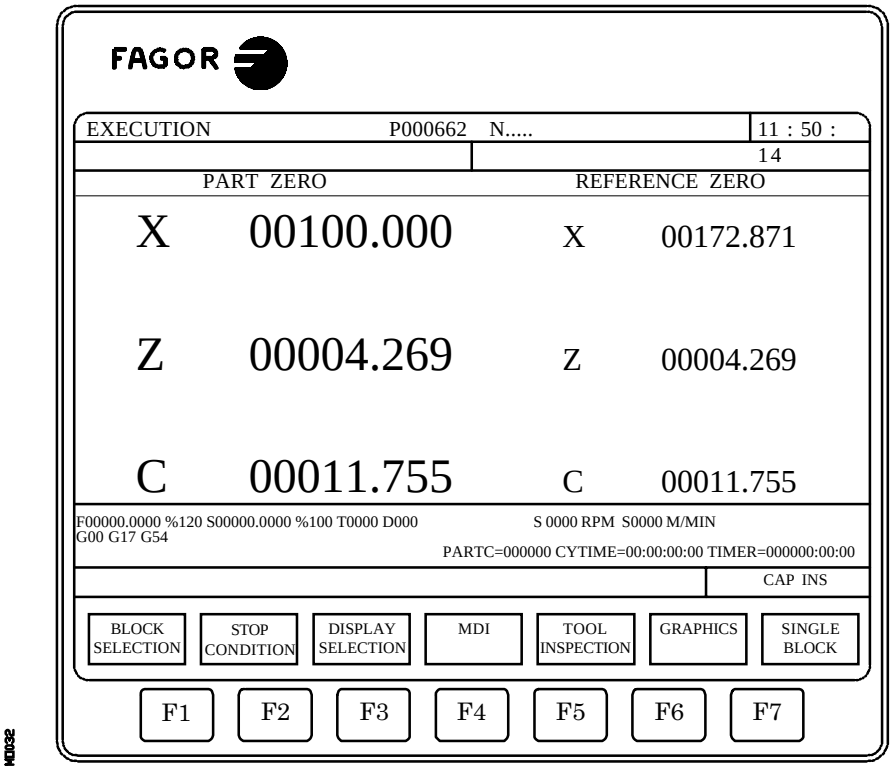
ACTUAL. Indicates the actual (current) position of the axis.

TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.2.2 POSITION DISPLAY MODE

This display mode shows the position values of the axes.

This display mode shows the following fields or windows:



* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and the format defined with the axis machine parameter "DFORMAT".

Each axis has the following fields:

PART ZERO This field shows the real axis position with respect to part zero.

MACHINE ZERO This field shows the real axis position with the respect to machine reference zero (home).

3.2.3. PART PROGRAM DISPLAY MODE

Displays a page of program blocks among which the block being executed is highlighted.

3.2.4. SUBROUTINE DISPLAY MODE

This display mode shows information regarding the following commands:

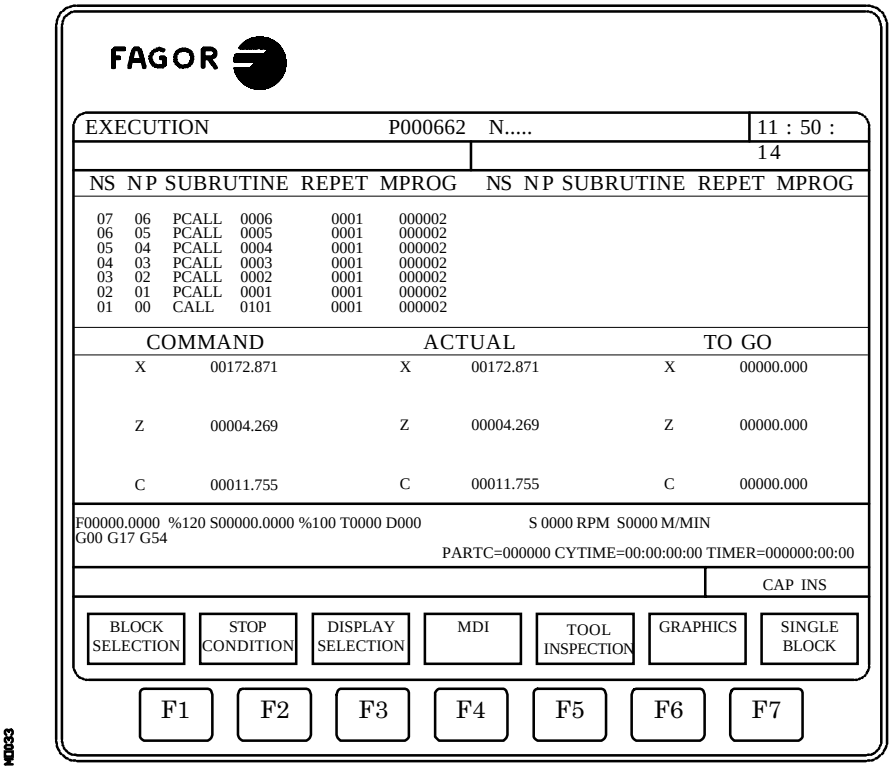
(RPT N10,N20)	This function executes the program section between blocks N10 thru N20.
(CALL 25)	This function executes subroutine number 25.
G87 ...	This function the corresponding canned cycle.
(PCALL 30)	This function executes subroutine 30 in a local parameter level.

When this mode is selected, the following must be considered:

The CNC allows the definition and usage of subroutines which can be called upon from a main program or from another subroutine and this can, in turn, call upon a second one and so forth up to 15 nesting levels (each subroutine call represents a nesting level).

The machining canned cycles G81 thru G89 use the next nesting level for local parameters when they are active.

This display mode shows the following fields or windows:



* Information on the subroutines which are active.

  NS Indicates the nesting level (1-15) which the subroutine occupies.

  NP Indicates the level of local parameters (1-6) in which the subroutine is executed.

  SUBROUTINE Indicates the type of block which has caused a new nesting level.

  Examples: (RPT N10,N20) (CALL 25) (PCALL 30) G87

  REPT Indicates the number of times which remain to be executed.

  For example, if (RPT N10, N20) N4 is programmed and is the first time that it is being executed, this parameter will show a value of 4.

  M If an asterisk is shown (*) this indicates that a Modal subroutine is active in this nesting level, and this is executed after each movement.

  PROG Indicates the program number where the subroutine is defined.

* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and in the format determined by the axis machine parameter "DFORMAT".

  Each axis has the following fields or windows:

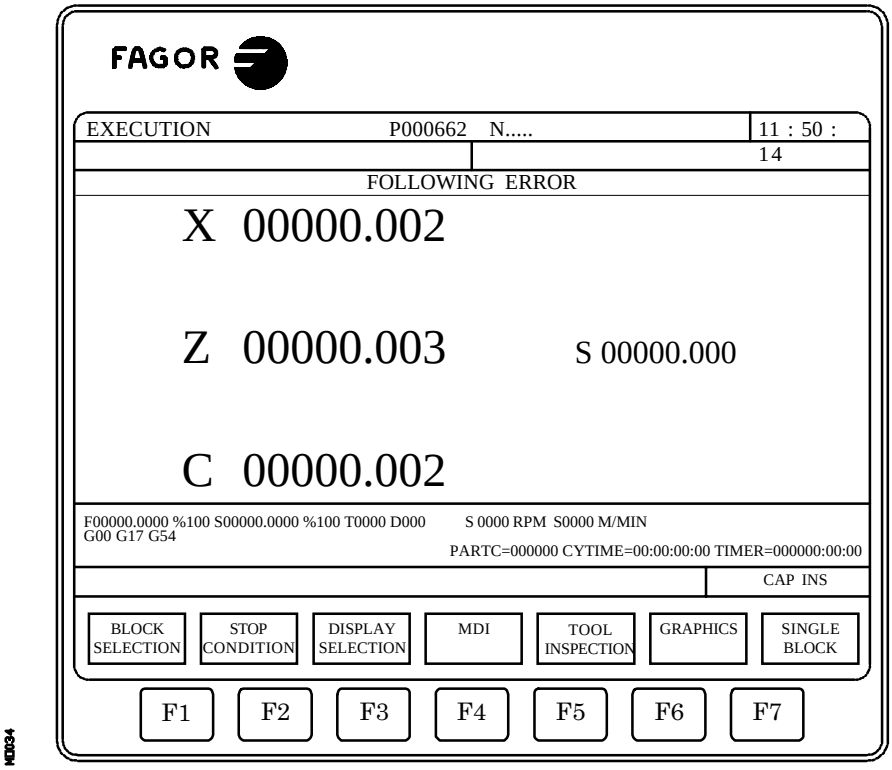
  COMMAND. Indicates the programmed coordinate or position which the axis must reach.

  ACTUAL. Indicates the actual (current) position of the axis.

  TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.2.5 FOLLOWING ERROR DISPLAY MODE

This display mode shows the following error (difference between the theoretical value and the real value of their position) of the axes and the spindle.



Note that the display format is determined by the axis machine parameter “DFORMAT”.

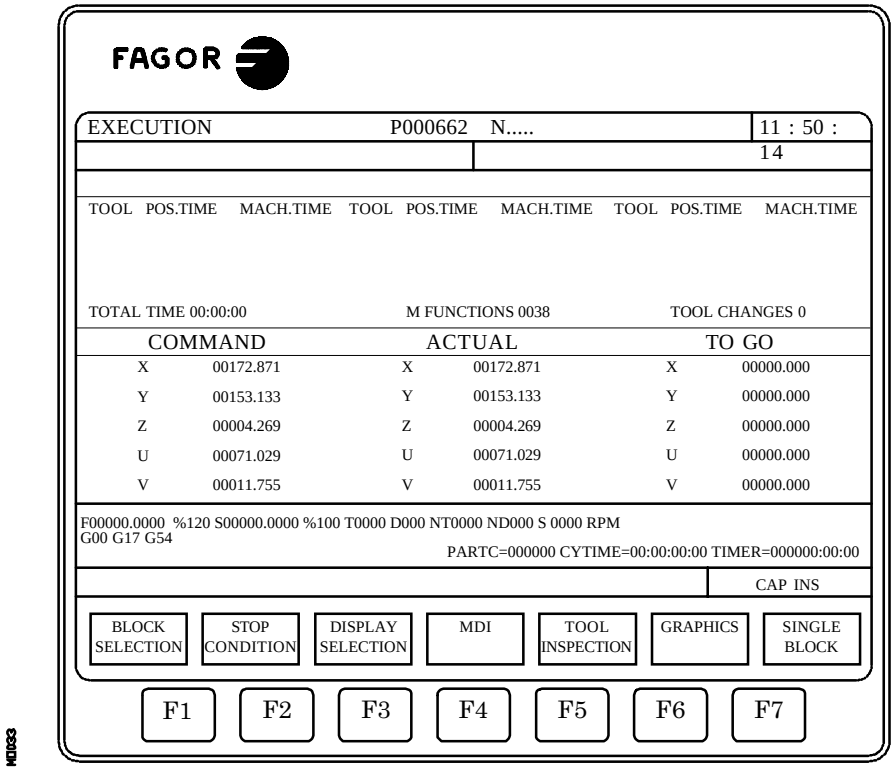
3.2.6 USER DISPLAY MODE

This option will execute the program which is selected by means of the general machine parameter “USERDPLY” in the user channel.

To quit this mode and return to the previous menu, press **ESC**.

3.2.7 EXECUTION TIME DISPLAY MODE

This option is available while simulating a part-program and it will display the following fields or windows:



* A display window shows the estimated program execution time at 100% of the programmed feedrate.

This display area shows the following information:

The time each tool (TOOL) takes to execute the positioning moves (POS.TIME) as well as the machining moves (MACH.TIME) indicated in the program.

The "TOTAL TIME" required to execute the complete program.

The "M FUNCTIONS" being exectued in the program.

The number of "TOOL CHANGES" performed during the execution of the program.

* The position values for the axes of the machine.

It must be borne in mind that the display format for the axes is established by machine parameter "**DFORMAT**" and that real or theoretical position values will be shown depending on the setting of machine parameter "**THEODPLY**".

Each axis has the following fields:

COMMAND. Indicates the programmed coordinate or position which the axis must reach.

ACTUAL. Indicates the actual (current) position of the axis.

TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.3 MDI

This function is not available in the SIMULATION mode. Besides, if a program is being executed, it must be interrupted in order to access this function.

It is possible to execute any block (ISO or high level) and it provides information on the corresponding format via the softkeys.

Once the block has been edited and after the key has been pressed the CNC will execute this block without quitting this operating mode.

3.4 TOOL INSPECTION

This function is not available in the SIMULATION mode. Besides, if a program is being executed, it must be interrupted in order to access this function.

This operating mode allows all the machine movements to be controlled manually, and enabling the axis control keys on the Operator Panel (X+, X-, Y+, Y-, Z+, Z-, 4+, 4-, etc.).

Also, the CNC will show the softkeys to access the CNC tables, edit and execute a block in MDI as well as repositioning the axes of the machine to the position from where this function was called.

One of the ways to make the tool change is as follows:

* Move the tool to the required tool change position

  This move may be made by jogging the axes from the operator panel or in MDI.

* Gain access to CNC tables (tools. Tool offsets, etc.) in order to find another tool with the similar characteristics.

* Select, in MDI, the new tool as the active one.

* Make the tool change

  This operation will be performed depending on the type of tool changer used. It is possible to execute the tool change in MDI in this step.

* Return the axes to the position where the tool inspection began (REPOSITIONING).

* Resume program execution ( )

Note: If during tool inspection, the spindle is stopped, the CNC will start it again when repositioning in the same direction as it was turning before (M3 or M4).

The CNC offers the following options by means of softkeys:

MDI

Allows to edit blocks in ISO or high level (except those associated with subroutines) providing information on the corresponding format by means of softkeys.

Once the block has been edited and after the key has been pressed the CNC will execute this block without quitting this operating mode.

TABLES

Allows access to any of the CNC tables associated with part programs (Zero offsets, Tool offsets, Tools, Tool magazine, Global and Local Parameters).

Once the desired table has been selected, all editing commands will be available for its verification and modification.

In order to return to the previous menu the **ESC** key must be pressed.

REPOSITIONING.

Positions the axes at the point where tool inspection started.

To do this, one of the following softkeys must be selected:

[PLANE] It moves the X and Z axes at the same time.
[X-Z] It moves first the X axis and then the Z axis.
[Z-X] It moves first the Z axis and then the X axis.

Then, press for the CNC to reposition the axes.

3.5 GRAPHICS

With this function it is possible to select the type of graphic to be used as well as to define all the parameters for the corresponding graphic display.

To do so, the CNC must NOT be executing or simulating a part program; otherwise, it must be interrupted.

Once the type of graphics has been selected and its parameters defined, this function can be accessed even during the execution or simulation of a part program should the type of graphic or any graphic parameters be changed

After selecting this function, the CNC will display the following softkey options:

* Type of graphic
* Display area
* Zoom
* Graphic parameters
* Clear Screen
* Deactivate graphics

3.5.1 TYPE OF GRAPHICS

The CNC has two types of graphics, line graphics and solid graphics.

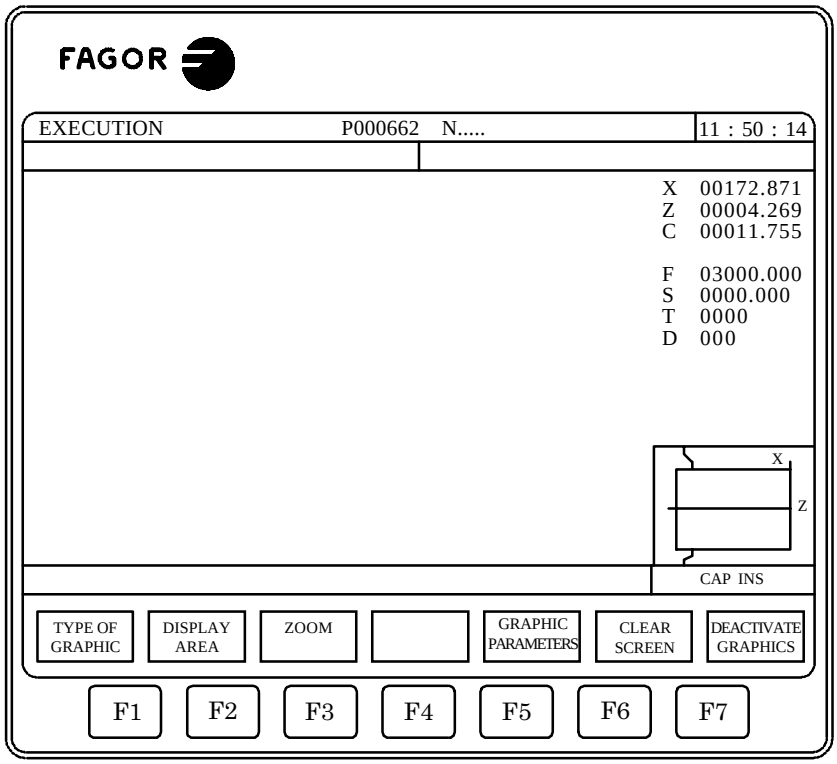
Both type options will be shown by means of softkeys in order to select one of them.

The selected type will remain active until a different type is selected, until graphics are deactivated or until the CNC is turned off.

Every time a type of graphics is selected, all the graphic conditions (zoom, graphic parameters and display area) used last will be recovered. These conditions remain even after turning the CNC off.

The coordinate system used in the graphic display is determined by the general machine parameter “GRAPHICS”.

The type of graphic selected will appear to the right of the screen with the following information:



TD035

* The theoretical cutter's point coordinates (X,Z)

* Feedrate and spindle speed currently selected.

* The coordinate system used in the graphic display which corresponds to the one set at general machine parameter "GRAPHICS".

* A clamped part with a superimposed window.

  The part shows the size of the display area and the superimposed window represents the area actually selected for the graphic display.

LINE GRAPHICS. XZ, XC, ZC

This type of graphics draws with colored lines the movement of the tool on the selected planes (XZ, XC, ZC).

The generated graphic will be lost in the following cases:

* When deleting the screen ("CLEAR SCREEN" softkey)
* When deactivating the graphics ("DEACTIVATE GRAPHICS" softkey).
* When redefining a new display area ("DISPLAY AREA" softkey).
* When selecting another type of solid graphics (solid XZ, solid XC, solid ZC).

SOLID GRAPHICS

This type of graphics shows the part secured at the spindle and the tool in use.

The types of solid graphics available are: XZ solid, XC solid and ZC solid.

Depending on whether the program is being executed or simulated, the programmed tool movement will be shown as well as the shape of the part after that operation.

If during the execution or simulation of a program, no tool is selected, the CNC will NOT show the tool and the shape of the part will be shown intact.

In **XZ SOLID**, when an "S" spindle speed is assigned, the resulting part will be shown machined (turned) on both sides. On the other hand, if no "S" is programmed, the resulting part will only be shown machined on the tool side.

The generated graphic will be lost in the following cases:

* When deleting the screen ("CLEAR SCREEN" softkey)
* When deactivating the graphics ("DEACTIVATE GRAPHICS" softkey).
* When redefining the new display area ("DISPLAY AREA" softkey).
* When selecting another type of line graphics (XZ, XC, ZC).

3.5.2 DISPLAY AREA

In order to use this function, no part program may be in execution or simulation. If this is the case, it must be interrupted.

With this option it is possible to re-define the size of the display area by assigning new maximum and minimum values for the Z axis and for the part radius (X axis). These position values are referred to part zero.

In order to re-define the display area, the CNC will show, to the right of the screen, several windows indicating the current display area dimensions.

Use the up and down arrow keys to select the window whose value is to be changed and key in the new value.

Once all the desired windows have been assigned new values, press **ENTER** to validate them.

To quit this mode without making any changes, press **ESC**.

In line graphics (XZ, XC, ZC) there is a softkey [optimum area] which redefines the display area that contains, in all planes, all the tool paths already executed.

3.5.3 ZOOM

In order to use this option, the CNC must not be executing or simulating a part-program. If so, it must be interrupted.

With this option, it is possible to enlarge or reduce the graphics display area.

When selecting this option, the CNC will show a window superimposed on the current graphics and another one over the drawing at the lower right-hand side of the screen. These new windows indicate the new display area being selected.

Use the [zoom +] and [zoom-] keys to either enlarge or reduce the size of the new display area and the arrow keys to move the zoom window around to the desired location on the screen.

Once the new display area has been defined, press **ENTER** to validate the new values.

Press **ESC** to quit this ZOOM mode without making any changes to the initial values.

Every time a Zoom is carried out in line graphics (XZ, XC, ZC), it redraws the machining executed up to that point. If the number of points to be redrawn exceeds the amount of memory reserved for it, only the last points will be redrawn and the older ones will be lost.

When zooming into a solid graphics, the drawing will be initialized showing a new unmachined 3D block.

3.5.4 GRAPHIC PARAMETERS

This function can be used any time, even during part program execution or simulation:

With this function it is possible to modify the simulation speed and the colors used to draw the tool paths.

The modifications made to any parameter are immediately assumed by the CNC and can be made during the execution or simulation of the part program.

The softkey options displayed by the CNC are:

SIMULATION SPEED

With this option it is possible to modify the percentage of the speed used by the CNC to execute the part programs in the simulation modes.

The CNC will display a window at the top right-hand side of the screen indicating the current % of simulation speed.

This value can be modified by using the right and left arrow keys. Once the desired value is selected, press **ENTER** to validate the new value.

Press **ESC** to quit this function without making any changes to this field.

It is also possible to change the simulation speed while it is redrawing after a zoom. This lets you check the machining of a particular operation.

PATH COLORS

With this option it is possible to modify the colors used to draw the various tool paths in the execution and simulation modes. They can only be used in line graphics XZ. The available parameters are:

The color for representing rapid moves
The color for representing path without compensation
The color for representing path with compensation
The color for representing threading

The CNC will show a series of windows for the definition of graphics parameters.

Among the various colors to choose from, there is a black or “transparent” one. If this one is chosen for a particular path, this path will not be displayed on the screen.

If any of them is to be modified, first select the corresponding window using the up and down keys and then use right and left arrow keys to select the desired color.

Once the desired colors have been selected, press **ENTER** to validate the new choices or **ESC** to ignore the changes and leave this function with the original values intact.

COLOR OF SOLID

With this option it is possible to modify the colors used to draw the solid. These values will be taken into account for part porgram simulation and execution. The following parameters are available:

Color for the cutter.
Color for the part.
Color for the axes.
Color for the clamps.

The CNC will show, to the right of the screen, a series of windows indicating the currently selected colors.

Among the various colors offered, it is also possible to select "black". If this color is selected to draw the part, none of the machining operations will be shown.

To modify any of them, use the up and down arrow keys to select the corresponding window and, then , the right and left arrow keys to select the desired color.

Once the colors have been selected, press **ENTER** for the CNC to assume the new values.

To quit this mode without altering the original values, press **ESC** instead.

3.5.5 CLEAR SCREEN

In order to use this function, no part program may be in execution or simulation. If this is the case, it must be interrupted.

Erases the screen or graphic representation shown.

If the solid graphic mode is selected, it will return to its initial status showing the unmachined part.

3.5.6 DEACTIVATE GRAPHICS

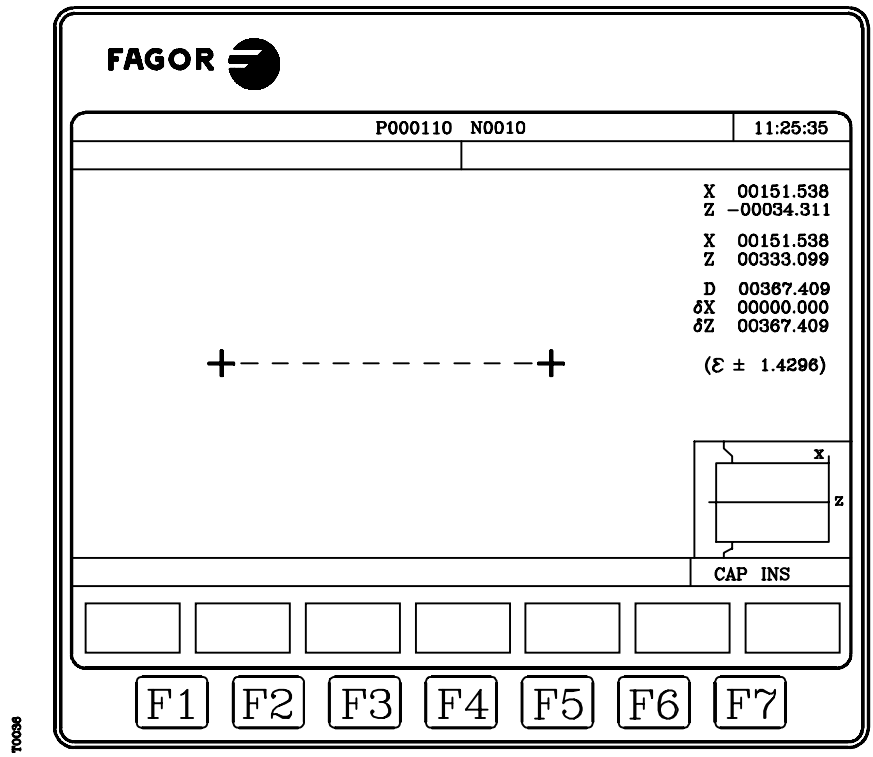
It allows the graphic representation to be deactivated at any time, even during execution or simulation of a part program.

To activate this function again, the “GRAPHICS” softkey must be pressed again. To do this, the CNC must not be executing or simulating a part program. If this is the case, it must be interrupted.

3.5.7 MEASURE

To use this function, a "Graphic" (planes XZ, XC or CZ) must be selected and the CNC must not be executing orsimulating the part-program. If it is, it must be interrupted.

Once this function is selected, the CNC shows the following information on the screen:



The center of the CRT shows a dotted line with two cursors, the section to be measured. Also, the right-hand side of the screen shows:

* The coordinates of those two cursors with respect to part-zero.

* The distance "D" between them and the components of this distance along the axes of the selected plane " X" and " Z".

* The cursor step " " corresponding to the selected display area. It is given in the work units, millimeters or inches.

The CNC shows the selected cursor and its coordinates in red.

To select the other cursor, press the "+" or "-" key. The CNC shows the new selected cursor and its coordinates in red.

To move the selected cursor, use the up, down, right and left arrow keys.

Also, with the keystroke sequences: Shift-Up arrow, Shift-Down arrow, Shift-Right arrow and Shift-Left arrow, it is possible to move the cursor to the corresponding end.

To quit this command and return to the graphics menu, press [**ESC**]

Also, if [I] is pressed, the CNC exits this work mode and returns to the graphics menu.

3.6 SINGLE BLOCK

Every time this option is selected, the CNC will toggle the active mode of operation, it being possible to do it at any time, even during the part program execution or simulation.

If the single block mode is selected, the CNC will only execute one line of the program every time the is pressed.

The upper window of the screen will show the selected mode of operation. If continuous execution, no message will appear and if SINGLE BLOCK, it will display the message: SINGLE BLOCK.

4. *EDIT*

This operating mode will be used to edit, modify or look at a part-program stored in the CNC's RAM memory.

To edit a part-program stored in the "Memkey Card" (CARD A) or in the hard disk (HD), it must be previously copied into RAM memory.

To edit a part-program, enter the program number (up to 6 digits) from the keyboard or by selecting it with the cursor from the CNC's part-program directory and then pressing **ENTER**.

Move the cursor on the screen line by line with the “up and down” arrow keys or page by page with the “page up” and “page down” keys.

Once the program number has been entered, the CNC will display the softkeys for the following options:

EDIT *(See section 4.1)*
To edit new lines in the selected program.

MODIFY *(See section 4.2)*
To modify an existing line of the program.

FIND *(See section 4.3)*
To search a string of characters within a program.

REPLACE *(See section 4.4)*
To replace a string of characters with another.

DELETE BLOCK *(See section 4.5)*
To delete a block or group of blocks.

MOVE BLOCK *(See section 4.6)*
To move a block or group of blocks within a program.

COPY BLOCK *(See section 4.7)*
To copy a block or group of blocks to another program position.

COPY TO PROGRAM *(See section 4.8)*
To copy a block or group of blocks into a different program.

INCLUDE PROGRAM *(See section 4.9)*
To insert the contents of another program into the one currently selected.

EDITOR PARAMETERS *(See section 4.10)*
To select the editing parameters (automatic numbering and axes for Teach-in editing).

4.1 EDIT

With this option it is possible to edit new lines or blocks of the selected program.

Select with the cursor the block after which the new ones will be added and press the softkey corresponding to one of the available editing modes.

CNC LANGUAGE ..... *(See section 4.1.1)*
The program is edited in ISO code or high level language.

TEACH-IN ..... *(See section 4.1.2)*
The machine is jogged to the desired position and, then, the new axis position may be assigned to the block.

INTERACTIVE ..... *(See section 4.1.3)*
Editing mode assisted by the CNC.

PROFILES ..... *(See section 4.1.4)*
To edit a new profile
After defining the known profile data, the CNC generates its corresponding ISO-coded program.

PROFILE SELECTION
To modify an existing profile.
The CNC requests the first and last blocks of the profile.
Once they are both defined, the CNC will show the corresponding graphics.
Section 4.1.4 describes how to operate with the profile.

USER
When selecting this option, the CNC will execute, in the user channel, the customizing program selected by general machine parameter "USEREDIT".*(See section 4.1.1)*
This is edited in ISO-code or high level language.

4.1.1 EDITING IN CNC LANGUAGE

A program will be edited block by block and each block can be written either in ISO code or high level language or it can be just a program comment.

Once this option has been selected, the softkeys will change colors and they will appear over white background showing the information corresponding to the type of editing possible at that point.

Also, editing help will be available at any time by just pressing the **HELP** key. To quit this help mode, press **HELP** again.

If ESC is pressed while editing a block, the block editing mode is abandoned and the block currently being edited will not be added to the program.

Once the block has been edited, press **ENTER.** This new block will be added to the program after the one indicated by the cursor.

The cursor will position over the new edited block and the editing area (window) will be cleared so another block can be written.

To quit the block editing mode, press **ESC** or **MAIN MENU.**

4.1.2 TEACH-IN EDITING

It is basically identical to the previous option (editing in CNC language), except what regards the programming of position coordinate values.

This option shows the current position values of each one of the axes of the machine.

It permits to enter the axes position values from the CNC keyboard (as when editing in CNC language) or, also, use the TEACH-IN editing format as described next.

* Jog the machine axes with the jogging keys or with the electronic handwheel up to the desired position.

* Press the softkey corresponding to the axis to be defined.

* The CNC will assign to this axis its current physical position as the program position value.

Either position value programming methods can be used at any time while defining a block.

When the block being edited has no information (empty editing area or window), the **ENTER** key may be pressed in which case the CNC will generate a new block with the current position values of the axes.

This block will be added automatically to the program and it will be inserted after the block indicated by the cursor.

The cursor will position over the new edited block and the editing area will be cleared so another can be written.

When the position values of <u>all</u> the axes are <u>not</u> to be programmed in this fashion, the CNC permits to select the desired axes. To do this, in this operating mode and within the "EDITOR PARAMETERS" option there is a soft key for "**TEACH-IN AXES**"

4.1.3 INTERACTIVE EDITOR

This editor leads the operator through the program editing process by means of questions he/she will answer.

This type of editing offers the following advantages:

* No knowledge of the CNC programming language is required.

* The CNC only admits the data it is requesting, thus no erroneous data can be entered.

* The programmer has, at all times, the appropriate programming aide by means of screens and messages.

When selecting this option, the CNC displays in the main window, a series of graphic options selectable by softkey.

If the selected option has more menus, the CNC will keep showing new graphic options until the desired one is selected.

From this moment, the information corresponding to this option will appear in the main window and it will start requesting the data necessary to program it.

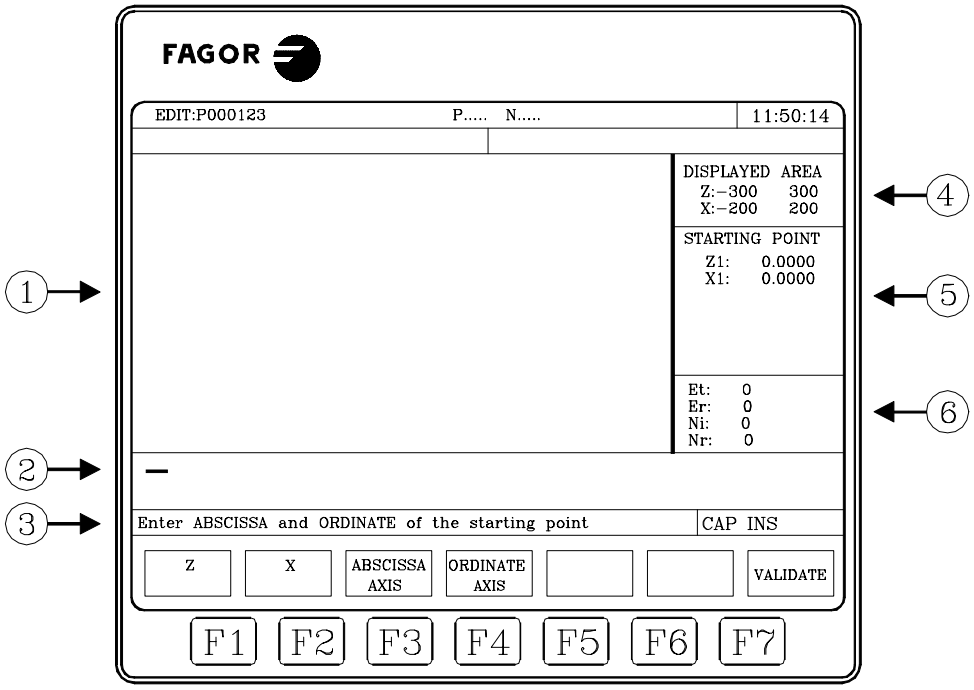
As the requested data is entered, the editing window will show, in CNC language, the block being edited.

The CNC will generate all necessary blocks and it will add them to the program once the editing of this option is done and it will insert them after the one indicated by the cursor.

The main window will show again the graphic options corresponding to the main menu being possible to continue editing the program.

4.1.4 PROFILE EDITOR

When selecting this option, the CNC displays the following fields or windows:



1.- Window showing the graphic representation of the profile being edited.

2.- Editing window showing the new generated block in CNC language.

3.- Area for editing messages.

4.- Display area

Indicates the area of the plane shown in the graphic representation of the profile. Indicated by the maximum and minimum position values of each axis.

The way to select this display are is described later on.

5.- Display area for the profile section currently selected for editing or modifying.

It may be the starting block, straight line, a clockwise arc or a counter-clockwise arc.

6.- Display area for additional information. It shows a series of parameters for internal use and whose meanings are:

Et : Total elements of the profile
Ec : Complete elements
Ni : Number of data entered
Nr : Number of required data

4.1.4.1 OPERATION WITH THE PROFILE EDITOR

Several profiles may be edited without quitting the profile editor. To edit a profile, proceed as follows:

1.- Select a point of the profile as its beginning point.

2.- Break the profile into straight and curve sections.

If the profile has corner roundings, chamfers, tangential entries or exits, take one of the following actions:

- Treat them as individual sections when having enough information to define them.

- Ignore them when defining the profile and, once done defining the whole profile, select the corners showing those characteristics and enter the corresponding radius value.

CONFIGURATION

Use the [abscissa axis] and [ordinate axis] softkeys to select the editing plane.

The Autozoom function indicates whether the CNC recalculates the graphics display area or not when the edited lines go beyond it.

PROFILE

For editing any profile.

CIRCLE

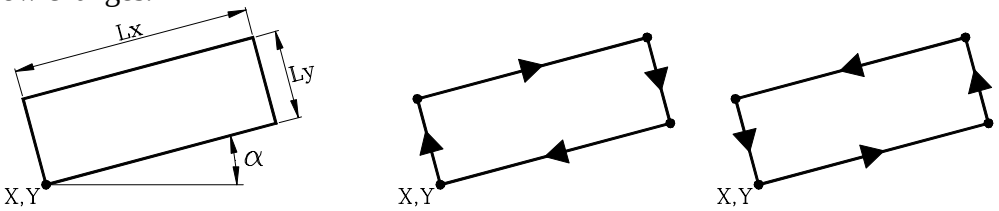
For a quick circular profile definition. If the starting point (X,Y) is not defined, the CNC assumes one.

The [Profile Direction] softkey indicates whether the profile is programmed clockwise or counterclockwise. This data is very important for later modifications and profile intersection. Every time this softkey is pressed, the text at the top of the middle right window changes.

STRAIGHTANGLE

For a quick straight angular profile definition.

The [Profile Direction] softkey indicates whether the profile is programmed clockwise or counterclockwise. This data is very important for later modifications and profile intersection. Every time this softkey is pressed, the text at the top of the middle right window changes.



clockwise counterclockwise

A straight angular profile is defined with a single command, but the CNC internally breaks into 4 straight segments.

4.1.4.2 PROFILE EDITING

When pressing the [PROFILE] softkey, the CNC requests the starting point of the profile. To define it, use the corresponding softkeys.

For example, if when working in the XY plane the new desired starting point is (20,50):

[X] 20 [ENTER]
[Z] 50 [ENTER]

The values may be set by means of a numeric constant or by means of any expression. Examples:

X 100
X 10 * cos 45
X 20 + 30 * sine 30
X 2 * (20 + 30 * sine 30)

Once the starting point has been set, press the [VALIDATE]

The CNC will show a filled circle in the graphics area to indicate the starting point of the profile.

Also, the softkeys will show the following options:

[STRAIGHT LINE]	To edit a straight section.
[CLOCKWISE ARC]	To edit a clockwise arc.
[COUNTERCLOCKWISE ARC]	To edit a counterclockwise arc.
[CORNERS]	To insert roundings, chamfers, tangential entries and exits.
[MODIFY]	To modify the starting point. Modify any profile element, even the type of element (straight line, clockwise or counterclockwise arc) Insert a new element (straight line or arc) in any position of the profile. Delete any profile element. Add a new additional text to any section of the profile. Modify the display area.
[NEW PROFILE]	To edit a new profile.
[FINISH]	It must be pressed when all the sections of the profile have been defined. It must be indicated whether the edited profile or profiles must be saved or not. The CNC quits the profile editor and adds to the program the ISO code corresponding to the profile just edited .

4.1.4.3 DEFINITION OF A STRAIGHT SECTION

When pressing the [STRAIGHT LINE] softkey, the CNC displays the data shown on the right margin of this page.

X1, Y1	Coordinates of starting point of the line. They cannot be modified because they correspond to the last point of the previous section.
X2, Y2	Coordinates of the end point of the section.
α	Angle of the line referred to the abscissa axis.
TANGENCY	Indicates whether the line to be drawn is tangent to the previous section or not.

All these parameters need not be defined, but all the known ones should be defined.

To define a parameter, press the corresponding softkey, key in the desired value and press [ENTER].

DISPLAY AREA

X: -300 300
Y: -200 200

STRAIGHT LINE

X1: 50.000
Y1: 60.000
X2:
Y2:
α :

TANGENCY: NO

Et: 0
Er: 0
Ni: 2
Nr: 2

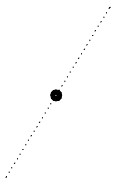
The value may be defined by a numeric constant or by any expression.

Examples:
X 100
X 10 * cos 45
X 20 + 30 * sine 30
X 2 * (20 + 30 * sine 30)

Once all known parameters are set, press the [VALIDATE] softkey and the CNC will show the defined section, if possible.

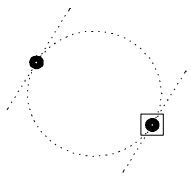
If there is not enough data to show the section, the CNC will show a dotted line indicating its orientation.

Example
X1=0
Y1=0
X2
Y2
α = 60



If there are more than one possibility, all the possible options will be shown and the desired one (framed in red) must be selected using the right and left arrow keys.

Example
X1
Y1
X2
Y2
α = 60
TANGENCY = YES



Use the up and down arrow keys to choose whether all the possible options are shown or only the one framed in red.

Once the desired option is selected, press [ENTER] for the CNC to assume it.

4.1.4.4 DEFINITION OF A CIRCULAR SECTION

When pressing the [CLOCKWISE ARC] or [COUNTER-CLOCKWISE ARC] softkey, the CNC displays the data shown on the right margin of this page.

X1, Y1	Coordinates of the starting point of the arc. They cannot be modified because they correspond to the last point of the previous section.
X2, Y2	Coordinates of the end point of the arc.
XC, YC	Coordinates of the arc center.
XC, YC	Radius of the arc.
TANGENCY	Indicates whether the arc to be drawn is tangent to the previous section or not.

All these parameters need not be defined, but all the known ones should be defined.

DISPLAY AREA

X: -300 300
Y: -200 200

CLOCKWISE ARC

X1: 50.000
Y1: 60.000
X2:
Y2:
XC:
YC:
RA
TANGENCY: NO

Et: 0
Er: 0
Ni: 2
Nr: 2

To define a parameter, press the corresponding softkey, key in the desired value and press [ENTER].

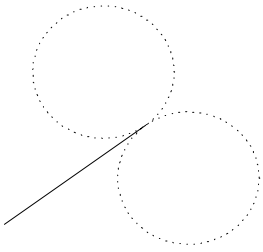
The value may be defined by a numeric constant or by any expression.

Examples:
X 100
X 10 * cos 45
X 20 + 30 * sine 30
X 2 * (20 + 30 * sine 30)

Once all known parameters are set, press the [VALIDATE] softkey and the CNC will show the defined section, if possible.

If there are more than one possibility, all the possible options will be shown and the desired one (framed in red) must be selected using the right and left arrow keys.

Example
X1 = 40
Y1 = 30
X2
Y2
XC
YC
RA = 20
TANGENCY = YES



Use the up and down arrow keys to choose whether all the possible options are shown or only the one framed in red.

If there is not enough data to show the section, the CNC waits for more data in order to solve the profile.

Once the desired option is selected, press [ENTER] for the CNC to assume it.

4.1.4.5CORNERS

When selecting this option, the CNC shows the following option softkeys:

Rounding	For rounding any corners of the profile.
Chamfer	For adding chamfers at any corner of the profile.
Tangential Entry	To add a tangential tool entry when machining.
Tangential Exit	To add a tangential tool exit at the end of the machining operation.

When selecting one of these, one of the corners of the profiles will appear highlighted.
To select another corner of the same profile, use the up/down and left/right arrow keys.
To select a corner of another profile, use the [page up] and [page down] keys.

To define the rounding, enter the rounding radius and press [ENTER].

To define the chamfer, enter the chamfer radius and press [ENTER].

To define the tangential entry, enter the radius of the path that the tool has to follow when doing a tangential entry and press [ENTER].

To define the tangential exit, enter the radius of the path that the tool has to follow when doing a tangential exit and press [ENTER].

To quit the CORNER mode, press [ESC].

4.1.4.6MODIFY

When selecting this option, the CNC shows the following softkey options:

Starting Point	To modify the starting point of the profile.
Modify element	To modify any element of the profile, even the type of element (straight lines, clockwise or counterclockwise arcs).
Insert element	To insert a new element (straight line or arc) in any position of the profile.
Delete element	To delete any element of the profile.
Additional Text	To add additional text to any section of the profile.
Configuration	To add a new editing plane or redefine the Autozoom option.
Display area	To change the display area.

When selecting one of these options, one of the profile elements will be highlighted.

To select another element of the same profile, use the up/down and left/right arrow keys.

To select an element of another profile, use the [page up] and [page down] keys.

Starting point

- Select the desired element. The CNC shows the values used to define it.
- Select the starting point of the desired profile. The CNC shows the values used to define it.
- Modify the desired values and press the [VALIDATE] softkey.

Modify element

- Select the desired element. The CNC shows the values used to define it.
- It is possible to: change the type of section (straight or arc), redefine the existing data, define a new data or delete an existing one.
- To delete data, press the softkey that defines it and press [ESC]
- Once the element has been modified, press the [VALIDATE] softkey. The CNC recalculates the new profile with the data used to define that section and the next one (tangency, angle, etc.).

Insert element

- Select the point, or corner, after which the new one is to be inserted.
- Select the type of section (straight or arc), define it and press the [VALIDATE] softkey.
- The CNC recalculates the new profile with the data used to define that section and the next one (tangency, angle, etc).

Delete element

- Select the element to be deleted and confirm the command.
- The CNC recalculates the new profile.

Additional text

- Select the desired element. The CNC shows the ISO code corresponding to that section in the editing area.
- Add the desired text. Functions F, S, T, D, M or program comments may be added.
- Press the [VALIDATE] softkey.

Display area

When selecting this option, the following softkey options are shown:

- [Zoom +] to enlarge the image on the screen.
- [Zoom -] to reduce the image on the screen.
- [Optimum area] to show the full profile on the screen.
- The display area may be moved around with the [left arrow], [right arrow], [up arrow] and [down arrow] keys.
- Press the [VALIDATE] softkey. The CNC updates the values indicated in the upper right-hand window (DISPLAYED AREA).

To quit the MODIFY mode, press [ESC].

4.1.4.7 FINISH

This softkey must be pressed once all the sections of the profile have been defined.

The CNC will try to calculate the requested profile by previously solving all the unknowns.

If it finds several possibilities for certain sections, the CNC will show them for each section and the desired option (framed in red) will have to be chosen using the right and left arrow keys.

Once the whole profile has been solved, the CNC will show the code of the part program currently being edited.

The ISO-coded program for the edited profile is contained between these lines:

```
;************************** START **********************

;************************** END **********************
```

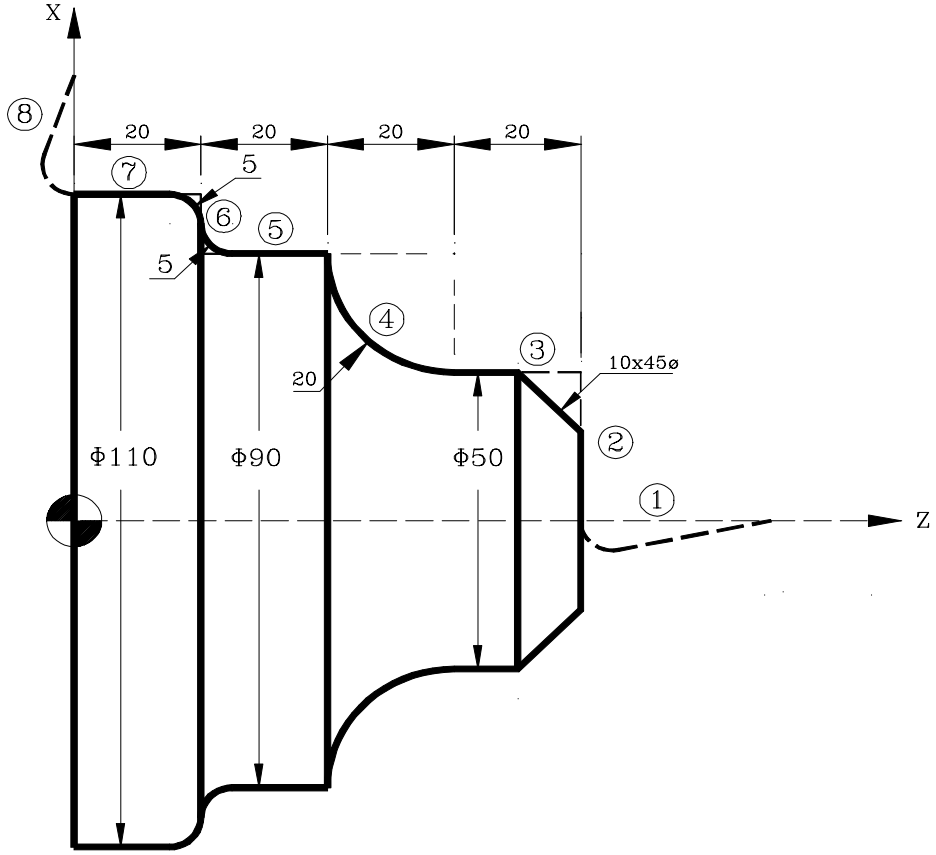
If a profile cannot be solved due to lack of data, the CNC will issue the corresponding error message.

Warning:

When pressing the [FINISH] softkey, the CNC quits the profile editor and adds to the program the ISO-code corresponding to the profile just edited.

To quit the profile editor without changing the part-program, press [ESC] and the CNC will request confirmation of this command.

4.1.4.8 EXAMPLES OF PROFILE DEFINITION



Profile definition without rounded corners, chamfers, tangential entries or exits

Abscissa and ordinate of the starting point		Z = 100	X = 0			
Section 1	STRAIGHT LINE	Z = 80	X = 0			
Section 2	STRAIGHT LINE	Z = 80	X = 50			
Section 3	STRAIGHT LINE	Z = 60	X = 50			
Section 4	CLOCKWISE ARC	Z = 40	X = 90	Zc = 60	Xc= 90	R= 20
Section 5	STRAIGHT LINE	Z = 20	X = 90			
Section 6	STRAIGHT LINE	Z = 20	X = 110			
Section 7	STRAIGHT LINE	Z = 0	X = 110			
Section 8	STRAIGHT LINE	Z = 0	X = 150			

Adapt the image to the screen

Select the "DISPLAY AREA" option and press the [OPTIMUM ZONE].

Definition of rounded corners, chamfers, tangential entries and exits.

Select the MODIFY option and define:

TANGENTIAL ENTRY		
	Select corner 1-2 and press [ENTER]	Set Radius = 5
CHAMFER	Select corner 2-3 and press [ENTER]	Set Radius = 10
ROUNDING	Select corner 5-6 and press [ENTER]	Set Radius = 5
ROUNDING	Select corner 6-7 and press [ENTER]	Set Radius = 5
TANGENTIAL EXIT		
	Select corner 7-8 and press [ENTER]	Set Radius = 5

Press [ESC] to quit the Modify option.

End of editing mode

Press the [FINISH] softkey. The CNC quits the profile editing mode and shows the generated program in ISO code.

4.2 MODIFY

This option permits modifying the contents of a selected program block.

Before pressing this softkey, select with the cursor the block to be modified .

Once this option is selected, the softkeys will change their color showing their type of modifying option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

By pressing **ESC**, the information corresponding to that block and which was shown in the editing area will be cleared. It will then be possible to modify its contents again.

To quit the block modifying mode, press **CL** or **ESC** to clear the editing window and then press **ESC** again. This way, the selected block will not be modified.

Once the block contents have been modified, press **ENTER** so the new contents replace the old ones.

4.3 FIND

This option is used to find a specific text within the selected program.

When selecting this option, the softkeys will show the following options:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

When the text is defined, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode. The cursor will be positioned where the indicated text was found last.

LINE NUMBER After pressing this key, the CNC requests the number of the block to be found. After keying in the desired number and pressing **ENTER**, the cursor will position over that block which will then be selected quitting the search mode.

4.4 REPLACE

With this function it is possible to replace a character sequence with another throughout the selected program.

When selecting this option, the CNC requests the character sequence to be replaced.

Once the text to be replaced is indicated, press the "WITH" softkey and the CNC will request the character sequence which will replace the previous one.

Once this text is keyed in, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of the searched text.

The search will begin at the current block.

The found text will be highlighted and the following softkey options will appear:

REPLACE Will replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

DO NOT REPLACE Will not replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

TO THE END This function will automatically replace all the matching text from the current block to the end of the program without offering the option of not replacing it.

ABORT This function will not replace the highlighted text and it will quit the "find and replace" mode.

4.5 DELETE BLOCK

With this function it is possible to delete a block or group of blocks.

To delete only one block, just position the cursor over it and press **ENTER**.

To delete a group of blocks, indicate the first and last blocks to be deleted. To do so, follow these steps:

* Position the cursor over the first block to be deleted and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be deleted and press the “FINAL BLOCK” softkey.

  If the last block to be deleted is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to delete them.

4.6 MOVE BLOCK

With this option it is possible to move a block or group of blocks by previously indicating the first and last blocks to be moved. To do so, follow these steps:

* Position the cursor over the first block to be moved and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be moved and press the “FINAL BLOCK” softkey.

  If the last block to be moved is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

  To move only one block, the “initial block” and the “final block” will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to move them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the “START OPERATION” softkey to carry out the move.

4.7 COPY BLOCK

With this option it is possible to copy a block or group of blocks by previously indicating the first and last blocks to be copied. To do so, follow these steps:

* Position the cursor over the first block to be copied and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be copied and press the “FINAL BLOCK” softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

  To copy only one block, the “initial block” and the “final block” will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to copy them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the “START OPERATION” softkey to carry out this command.

4.8 COPY A PROGRAM

With this option it is possible to copy a block or group of blocks of one program into another program.

When selecting this option, the CNC will request the number of the destination program where the selected block or blocks are to be copied. After entering the program number press **ENTER**.

Next, indicate the first and last blocks to copy by following these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks and will execute the command.

  If the destination program already exists, the following options will be displayed:

* Write over the existing program. All the blocks of the destination program will be erased and will be replaced by the copied blocks.

* Append (add) the copied blocks behind the ones existing at the destination program.

* Abort or cancel the command without copying the blocks.

4.9 INCLUDE A PROGRAM

With this option it is possible to include or merge the contents of another program into the one currently selected.

Once this option is selected, the CNC will request the number of the source program to be merged. After keying in that number press **ENTER**.

Next, indicate with the cursor the block after which the source program will be included.

Finally, press the “START OPERATION” softkey to execute the command.

4.10 EDITOR PARAMETERS

With this option it is possible to select the editing parameters used in this operating mode.

The options or parameters available are described here and they are selected by softkeys.

4.10.1 AUTONUMBERING

With this option it is possible to have the CNC automatically number (label) the blocks after the one being edited.

Once this option is selected, the CNC will display the "ON" and "OFF" softkeys to either activate or deactivate this function.

Once this function is activated, the following options will appear on the CRT:

STEP After pressing this softkey, Enter the desired numbering step between two consecutive blocks and press **ENTER**.

The default value is 10.

STARTING After pressing this softkey, Enter the starting block number to be used on the next block to be edited.

The default value is 0.

When setting both parameters, select the STEP first and then the STARTING block number.

Example: STEP = 12, STARTING= 56;
generated blocks: N56, N68, N80,...

Warning:

This function will not number the already existing blocks.

4.10.2 AXES SELECTION FOR TEACH-IN EDITING

Remember that in the TEACH-IN editing mode, the following feature is available:

When the block being edited has no information (editing area empty), the "**ENTER**" key can be pressed. In this case, the CNC will generate a new block with the current position values of the axes.

The option described here, permits the selection of the axes whose position values will be automatically entered in said block.

After pressing the "**TEACH-IN AXES**" softkey, the CNC shows all the axes of the machine.

The operator must eliminate, pressing the corresponding softkeys, the axis or axes not desired. Every time a softkey is pressed, the CNC will eliminate the corresponding axis displaying only the selected ones.

To end this operation, press "**ENTER**".

The CNC will assume from now on and whenever editing in TEACH-IN, the selected axes. To change those values, access this option again and select the new axes.

5. *JOG*

This mode of operation will be used whenever the manual control of the machine is desired.

Once this mode of operation is selected, the CNC allows the movement of all the axes by means of the axes control keys (X+, X-, Z+, Z-, 3+, 3-, 4+, 4-) located on the operator panel, or by means of the electronic handwheel (if available).

This mode of operation offers the following softkey options:

With the **MDI** option it is possible to modify the machining conditions (type of moves, feedrates, etc.) being selected. Also, the CNC will maintain the ones selected in this mode when switching to **"EXECUTION"** or **"SIMULATION"** modes.

This operating mode offers the following softkey options:

REFERENCE SEARCH

With this option it is possible to perform a home search on the desired axis or axes.

The CNC offers two ways to search the machine reference (home):

* Using the subroutine associated with function G74. The number of this subroutine will defined by the general machine parameter "REFSUB".

* By selecting the axis or axes to be referenced.

Once the Reference search function is selected, the CNC will show a softkey for each axis and the softkey "ALL".

If the "ALL" softkey is selected, the CNC will highlight (in reverse video) the names of all axes and after pressing the key, it will execute the subroutine associated with G74.

On the other hand, to search the reference anywhere from one to all axes at once (without executing the associated subroutine), the softkeys corresponding to those axes must be pressed.

After pressing each softkey, the CNC will highlight the name of the selected axis.

If an unwanted axis has been selected, press **ESC** to cancel that selection and return to select "REFERENCE SEARCH".

Once all the desired axes have been selected, press .

The CNC will start the home search by moving all selected axes at once until the home reference switches for all axes are pressed and, from then on, the CNC will continue the home search one axis at a time.

Warning:

When searching home using the "ALL" softkey, the CNC will maintain the part zero or zero offset active at the time. However, if the axes have been selected one by one, the CNC will assume the "home" position as the new part zero.

PRESET

With this function it is possible to preset the desired axis position value.

Once this option is selected, the CNC will show the softkey corresponding to each axis.

After pressing the softkey of the corresponding axis to be preset, the CNC will request the position value to be preset with.

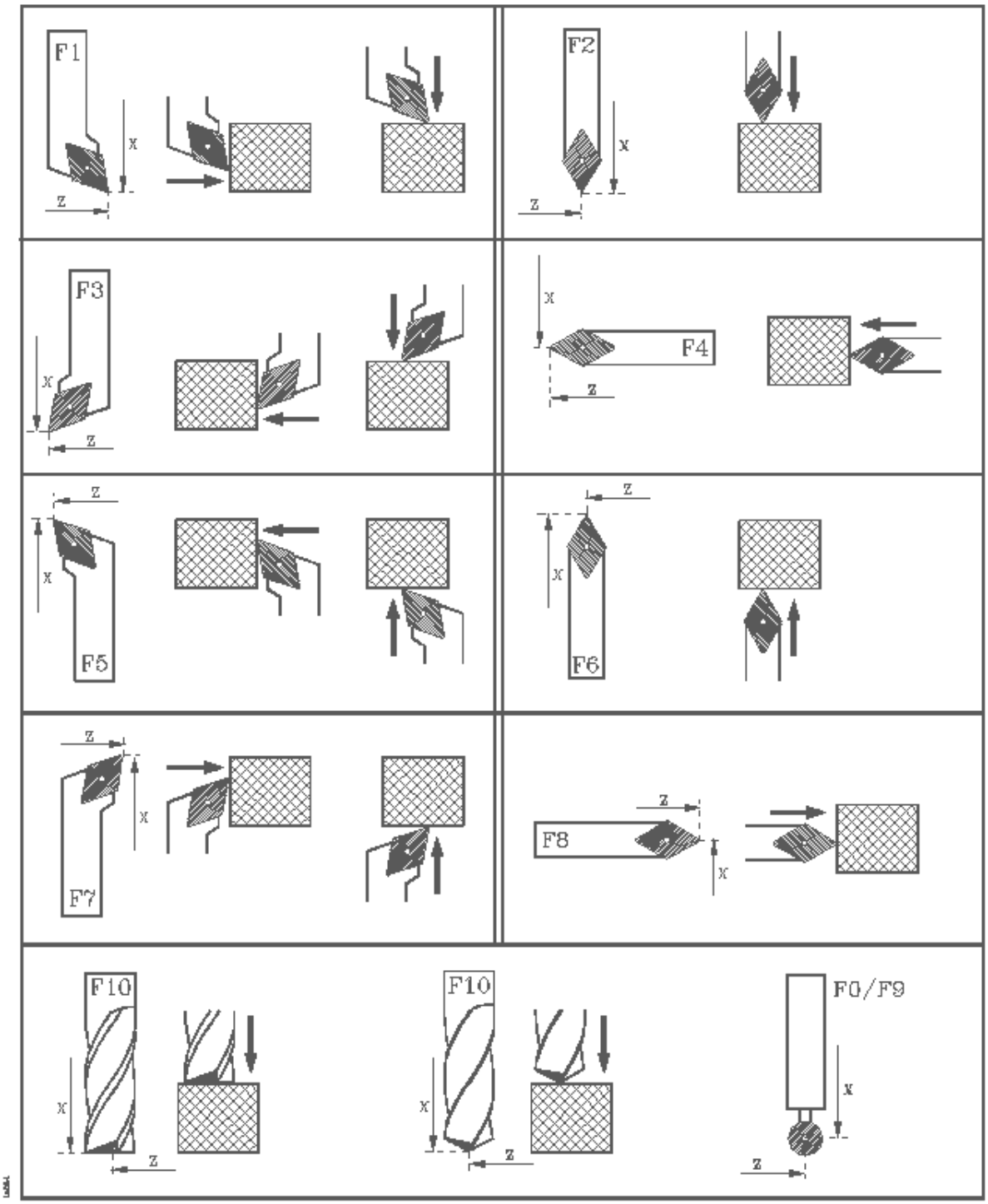
Press **ENTER** after the value has been keyed in so the new value is assumed by the CNC.

TOOL CALIBRATION

With this function it is possible to calibrate the lengths, along the X and Z axes, of the selected tool by using a part of known dimensions for this purpose.

Before pressing this softkey, the tool to be calibrated must be selected.

The tool calibration is done one axis at a time and in the following manner:



Tool calibration without a probe

The tool calibration is performed axis by axis (X, Z).

Follow these steps:

* Press the softkey corresponding to the axis [X] or [Z].

* The CNC will request the position value of the known part at the touch point.

  Once this value has been keyed in, press **ENTER** for this value to be assumed by the CNC.

* Jog the tool with the jog-keys (X+, X- Z+, Z-, 3+, 3-, 4+, 4-) until touching the part.

* Press the "LOAD" softkey corresponding to this axis.

  The CNC will perform the necessary calculations and it will assign the new value to the tool length.

Repeat these operation to calibrate the tool along the other axis.

Tool calibration with a probe

When using a probe for tool calibration, the following machine parameters must be properly set: **"PRBXMIN", "PRBXMAX", "PRBYMIN", PRBYMAX", "PRBZMIN" and "PRBZMAX**".

The tool calibration is performed axis by axis (X, Z) in either one of these modes: "without a probe" as described above and as follows:

Follow these steps:

* Position the tool in front of the probe.

* Press the softkey which indicates the direction of the tool calibration.
  [PROBE X+], [PROBE X-], [PROBE Z+], [PROBE Z-].

* The CNC will move the tool at the feedrate indicated by the machine parameter for that axis "**PRBFEED**" until touching the probe.

  The maximum distance the tool can move is set by machine parameter "**PRBMOVE**".

* When the tool touches the probe, the CNC stops the axis and, after making the pertinent calculations, it will assign the new tool length value to its corresponding tool offset.

Repeat these operation to calibrate the tool along the other axis.

MDI

With this function it is possible to edit and execute a block (ISO or high-level) providing the necessary information by means of softkeys.

Once the block has been edited, press [I] to execute it without leaving this operation mode.

Warning:

When searching home "G74", the CNC will maintain the part zero or zero offset active at the time.

USER

When selecting this option, the CNC will execute, in the user channel, the program whose number is indicated in the general machine parameter "USERMAN".

To quit its execution and return to the previous menu, press **ESC**.

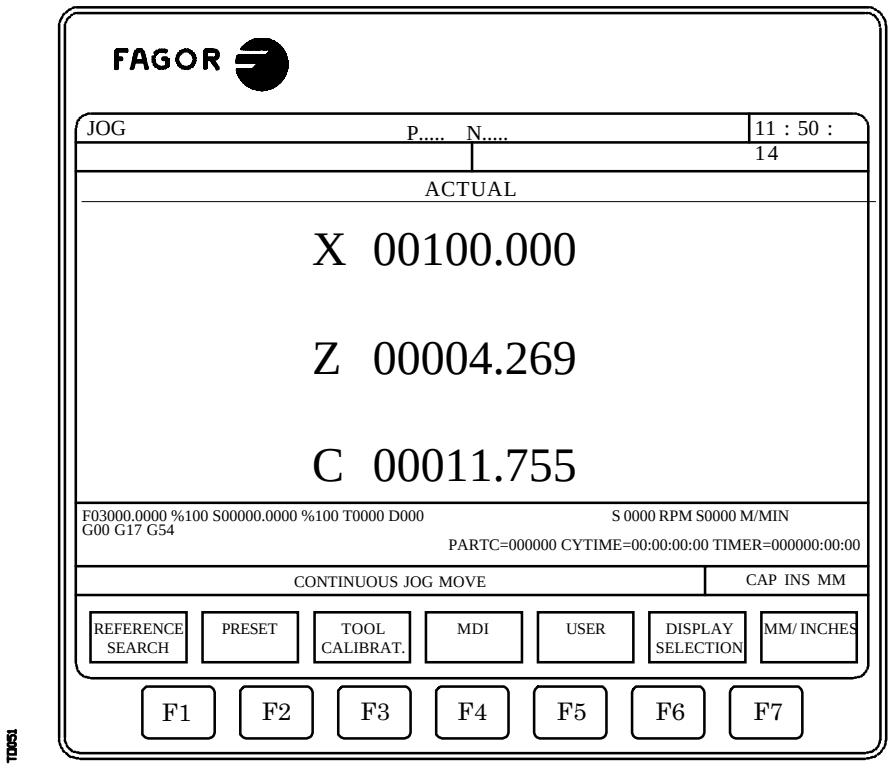
DISPLAY SELECTION

With this function it is possible to monitor the PLC by pressing the corresponding softkey. Once in that mode, operate as described in the chapter regarding the monitoring of the PLC.

It is also possible to select with the corresponding softkey one of the following position value (coordinate) displays:

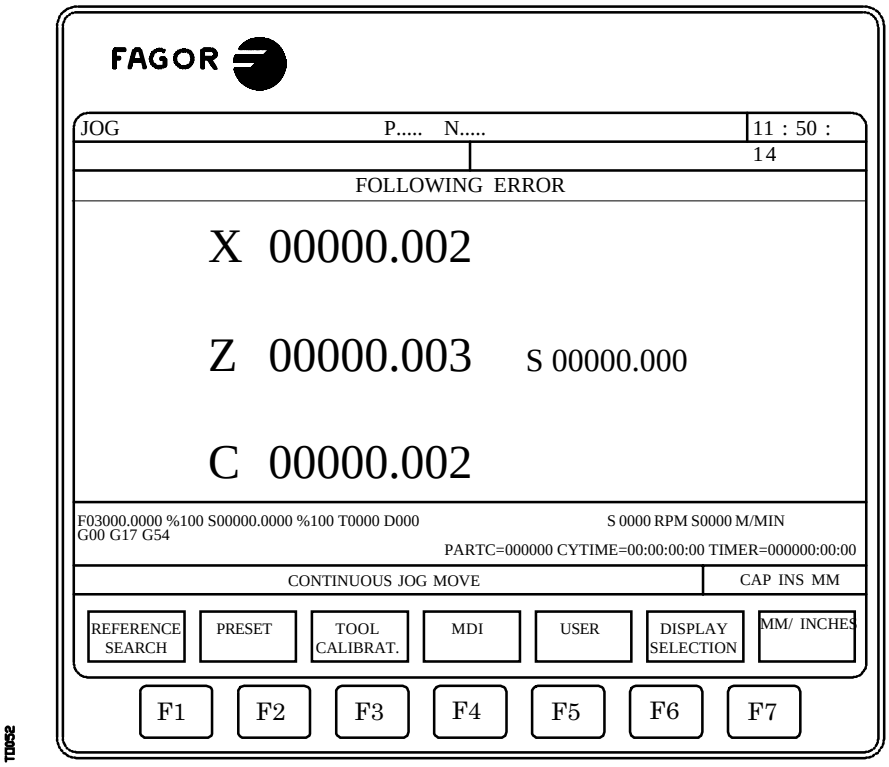
ACTUAL

When selecting this option, the CNC will show the current position of the axes with respect to part zero.



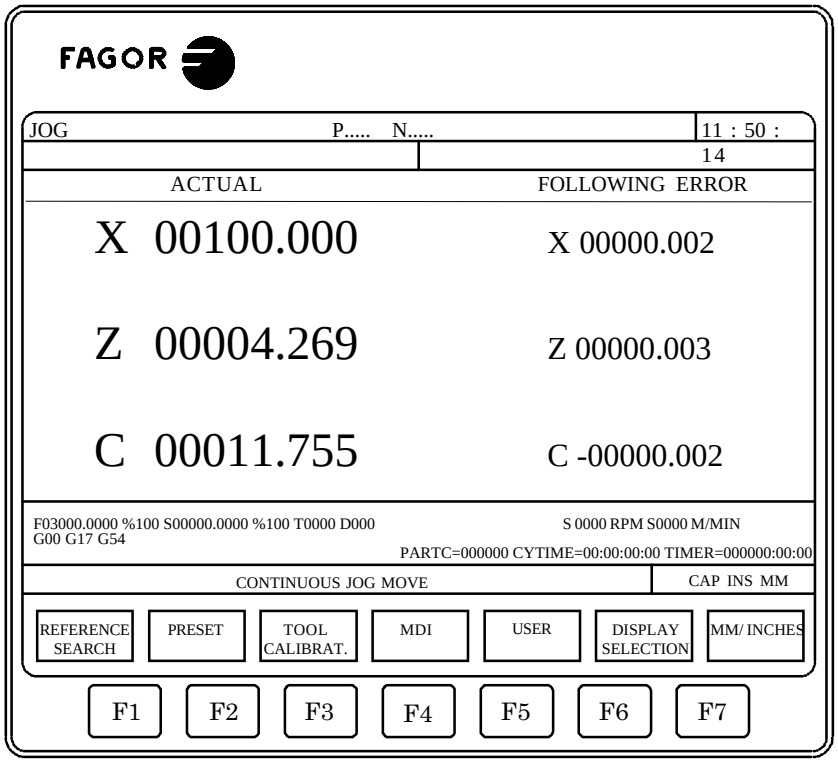
FOLLOWING ERROR

When selecting this option, the CNC will show the following error (difference between the theoretical and real positions of the axes) for each axis and the spindle.



ACTUAL AND FOLLOWING ERROR

When selecting this option, the CNC will show both the actual axes positions and their following errors.



TD053

MM/INCHES

This softkey toggles the display units for the linear axes from millimeters to inches and vice versa.

The lower right-hand window will indicate which units are selected at all times.

Note that this switching obviously does not affect the rotary axes which are shown in degrees.

5.1 JOGGING THE AXES

5.1.1 CONTINUOUS JOG

Once the % override of the jogging feedrate (indicated by axis-machine parameter "JOGFEED") has been selected with the switch at the Operator Panel , press the jog keys corresponding to the desired axis and to the desired jogging direction (X+, X-, Z+, Z-, 3+, 3-, 4+, 4- etc.).

The axes can be jogged one at a time and in different ways depending on the status of the general logic input "LATCHMAN":

* If the PLC sets this mark low, the axes will be jogged while pressing the corresponding Jog key.

** If the PLC sets this mark high, the axes will be jogged from the time the corresponding Jog key is pressed until the key is pressed or another jog key is pressed. In this case, the movement will be transferred to the axis corresponding to the new jog key.

If while jogging an axis, the key is pressed, the axis will move at the feedrate established by machine parameter "G00FEED" for this axis as long as this key stays pressed. When releasing this key, the axis will recover the previous feedrate (with its override %).

5.1.2 INCREMENTAL JOG

It allows to jog the selected axis in the selected direction an incremental step selected by the Feedrate Override switch and at the feedrate indicated by machine Parameter for that axis "JOGFEED".

The available positions are: 1, 10, 100, 1000 and 10000 corresponding to display resolution units.

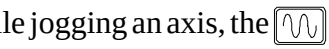
Example:

Display format: 5.3 in mm or 4.4 in inches

Switch position	Movement
1	0.001 mm or 0.0001 inch
10	0.010 mm or 0.0010 inch
100	0.100 mm or 0.0100 inch
1000	1.000 mm or 0.1000 inch
10000	10.000 mm or 1.0000 inch

The maximum permitted step is 10 mm or 1 inch regardless of the selected display format (for example: 5.2 in mm or 4.3 in inches).

After selecting the desired incremental move at the switch, if a jog key is pressed (X+, X-, Z+, Z-, 3+, 3-, 4+, 4- etc.), the corresponding axis will move the selected distance in the selected direction.

If while jogging an axis, the  key is pressed, the axis will move at a feedrate established by machine parameter "G00FEED" for this axis as long as this key stays pressed. When releasing this key, the axis will recover the previous feedrate (with its override %).

5.1.3 JOGGING WITH ELECTRONIC HANDWHEEL

The various handwheel configurations are:

General handwheel Is the typical handwheel. It can be used to jog any axis one by one. Select the axis and turn the handwheel to move it.

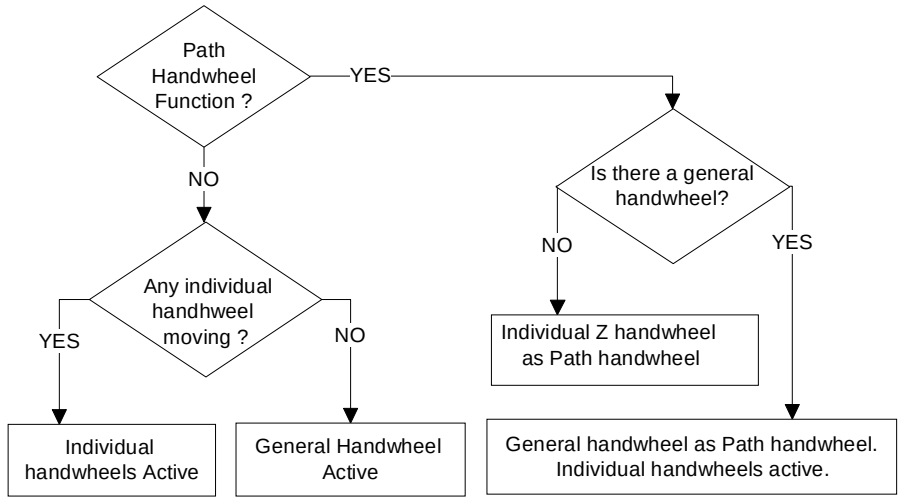
Individual handwheel: It replaces the mechanical handwheels. Up to 3 handwheels can be used (one per axis). It only moves the axis it is associated with.

Path handwheel: For chamfering and rounding corners. 2 axes are moved along a selected path by moving a single handwheel.

This feature must be activated via PLC.

The general handwheel is assumed as the "path handwheel" or, the individual handwheel associated with the Z axis

When using several handwheel types, the CNC sets the following priorities:



Operation when the "path handwheel" function is not active.

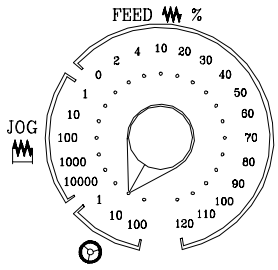
Any Individual handwheel has priority.
To jog with the general handwheel, select the axis and turn the handwheel.

Operation when the path handwheel function is active.

If there is no General Handwheel, the CNC assumes as path handwheel the individual handwheel associated with the Z axis.

If there is General handwheel, the CNC assumes it as path handwheel. The individual handwheels keep working.

To move any of them, turn the switch to any of the handwheel positions.



Positions 1, 10 and 100 indicate the multiplying factor being applied besides the internal x4 to the feedback pulses supplied by the electronic handwheel.

For example, when having a 25 lines/turn handwheel with a display format of 5.3 mm or 4.4 inches and the machine parameter "MPGRES=0":

Switch position	Distance per turn	
1	0.100 mm	or 0.0100 inch
10	1.000 mm	or 0.1000 inch
100	10.000 mm	or 1.0000 inch

To apply a different multiplying factor for each handwheel, the HANFCT variable must be used. Refer to the section on variables associated with electronic handwheels in chapter 10 of this manual.

Warning:

Depending on how fast the handwheel is turned and on the selected handwheel switch position, the CNC might be demanded to move the axis faster than the limit set by general machine parameter “G00FEED”. In that case, the CNC will move the axis the indicated distance but it will limit the axis speed to that parameter value.

5.1.3.1 GENERAL HANDWHEEL

Select the axis to be jogged

Press one of the JOG keys of the axis to be jogged. The selected axis will be highlighted.

When using a FAGOR handwheel with an axis selector button, the axis may be selected as follows:

Push the button on the back of the handwheel. The CNC select the first axis and it highlights it.

When pressing the button again, the CNC selects the next axis and so on in a rotary fashion.

To deselect the axis, hold the button pressed for more than 2 seconds.

Jog the axis

Once the axis has been selected, it will move as the handwheel is being turned and in the direction indicated by it.

5.1.3.2 INDIVIDUAL HANDWHEEL

Jog the axis

Each axis will move as the corresponding handwheel is being turned according to the switch position and in the direction indicated by it.

5.1.3.3 PATH HANDWHEEL

This feature allows to move two axes simultaneously with a single handwheel along a straight path or an arc for rounding or chamfering corners.

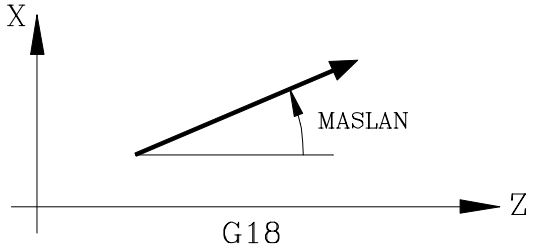
The CNC assumes as "Path Handwheel" the general handwheel or, when not available, the individual handwheel, the individual handwheel associated with the Z axis.

This feature must be handled by the machine manufacturer.

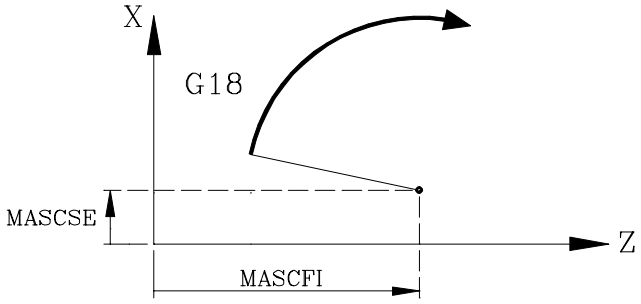
After slecting the "path handwheel" mode, the type of path must be indicated:

Straight line or an arc.

For a linear path, the path angle must be given at variable MASLAN (degrees between the linear path and the first axis of the plane)



For an arc, the center coordinates must be given at variables MASCFI and MASCSE (for the first and second axis of the main plane)



5.2 MANUAL CONTROL OF THE SPINDLE

It is possible to control the spindle by means of the following Operator-Panel keys without the need to execute M03, M04 or M05.

[C] is similar to executing M03. It starts the spindle clockwise and it displays M03 in the history of machining conditions.

[↻] is similar to Executing M04. It starts the spindle counter-clockwise and it displays M04 in the history of machining conditions.

[STOP] is similar to executing M05. It stops the spindle.

[+] and [−] vary the programmed spindle speed between the % set in spindle machine parameters “MINSOVR” and “MAXSOVR with incremental steps set in spindle machine parameter “SOVRSTEP”.

It is recommended to define the spindle speed before selecting the turning direction in order to avoid an abrupt start.

6. *TABLES*

In order to select a new tool, tool offset or zero offset, it is necessary that those values be previously stored at the CNC.

The tables available at the CNC are:

Zero Offset table. It must be defined. It indicates each axis offset for each zero offset.
Tool Offset table. It must be defined. It stores the dimensions of each tool.
Tool Table. It must be defined. It indicates for each tool, the family it belongs to, the offset associated to it, its nominal and real life spans, etc.
Tool Magazine Table. It must be defined. It indicates the position each tool occupies in the tool magazine.
Tool Geometry table. It must be defined. It indicates the dimensions and cutting angle of each cutter.
Global and local parameter table. It does not need to be defined. It is updated by the CNC.

When selecting a tool (T) or a tool offset (D), the CNC acts as follows:

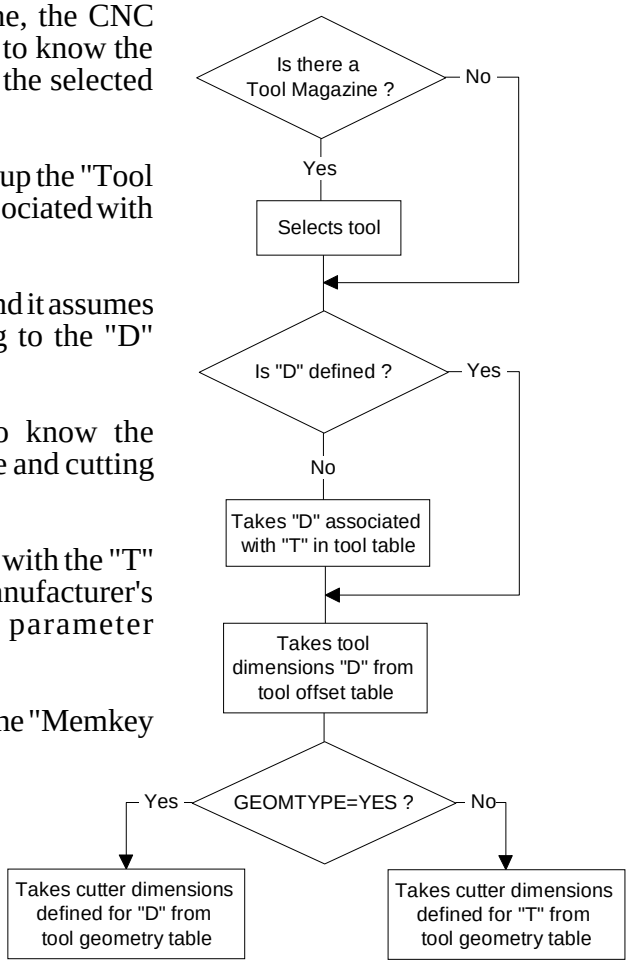
If the machine has a tool magazine, the CNC looks up the "tool magazine table" to know the position of the desired tool and of the selected tool.

If "D" has not been defined, it looks up the "Tool table" to know the tool offset "D" associated with it.

Examines the "Tool Offset Table" and it assumes the tool dimensions corresponding to the "D" offset.

Analyzes the "tool geometry" to know the geometry of the cutter (width, angle and cutting angle).

The "Geometry table" is associated with the "T" or with the "D" according to the manufacturer's criteria, general machine parameter "GEOMTYPE (P123)"

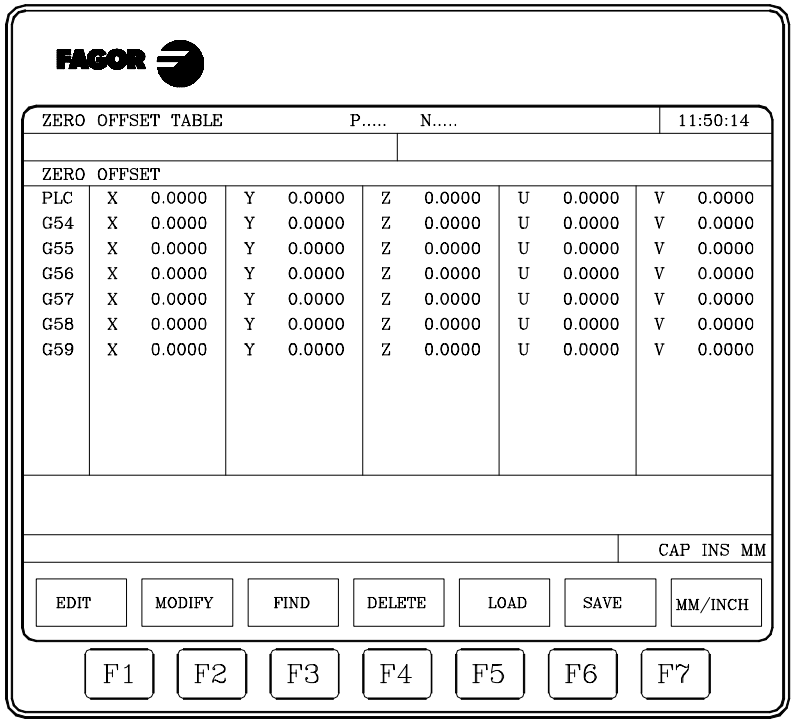

Is there a Tool Magazine ?
No
Yes
Selects tool
Is "D" defined ?
Yes
No
Takes "D" associated with "T" in tool table
Takes tool dimensions "D" from tool offset table
GEOMTYPE=YES ?
Yes
No
Takes cutter dimensions defined for "D" from tool geometry table
Takes cutter dimensions defined for "T" from tool geometry table

It is recommended to save the tables in the "Memkey Card" (CARD A) or out to a peripheral device or PC.

When accessing the TABLES operating mode, the CNC shows all the tables saved into the "Memkey Card" (CARD A).

6.1 ZERO OFFSET TABLE

This table stores the offset of each axis.



The end of the chapter describes how to edit the table. The possible zero offsets are

PLC. Additive zero offset defined by PLC.

It is used, among others, to compensate for possible deviations due to machine dilatation.

These values are set from the PLC and from the part-program, by means of high level variables "PLCOF(X-C)".

The CNC always adds these values to the zero offset currently active.

G54 through G57. Absolute zero offsets.

They can also be modified from the PLC and from the part-program, by means of high level variables "ORG(X-C)".

In order for one of these absolute zero offsets to be active, it must be selected at the CNC using its corresponding G code (G54, G55, G56 or G57).

G58 and G59. Incremental zero offsets.

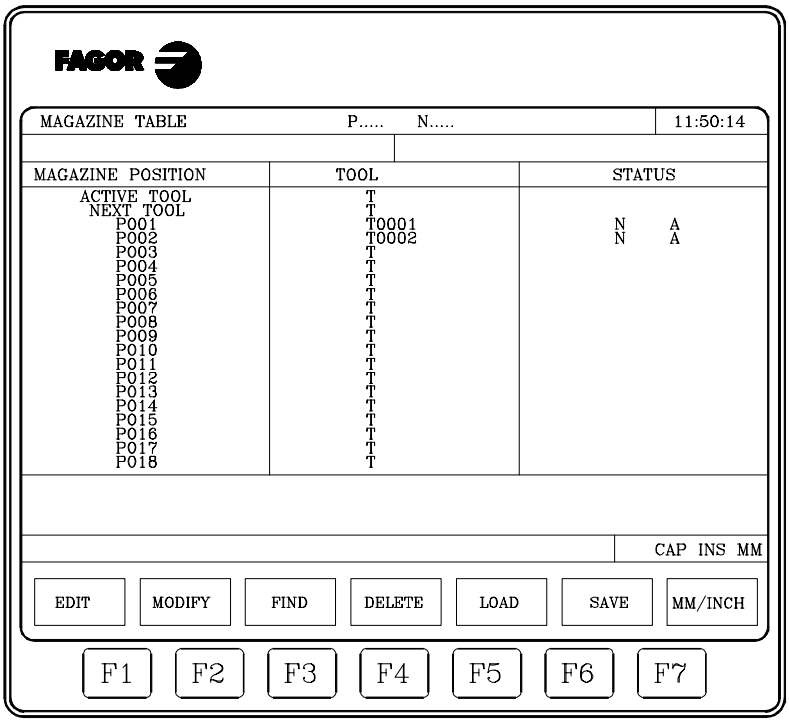
They can also be modified from the PLC and from the part-program, by means of high level variables "ORG(X-C)".

In order for one of these incremental zero offsets to be active, it must be selected at the CNC using its corresponding G code (G58 or G59).

The new incremental zero offset will be added to the absolute zero currently selected.

6.2 TOOL MAGAZINE TABLE

This table contains information about the tool magazine indicating all the tools of the magazine and their position in it.



The end of this chapter describes how to edit the table.

Magazine position

Besides indicating each position in the magazine, it indicates the active tool and the one selected for the following operations.
The next tool will be placed in the spindle after executing auxiliary function M06.

Tool

It indicates the number of the tool occupying that position.
The empty positions appear with the letter "T" and the canceled ones with the characters T****.

Status

The first letter indicates the tool size and the second one its status.

The size depends on the number of pockets it occupies in the magazine.

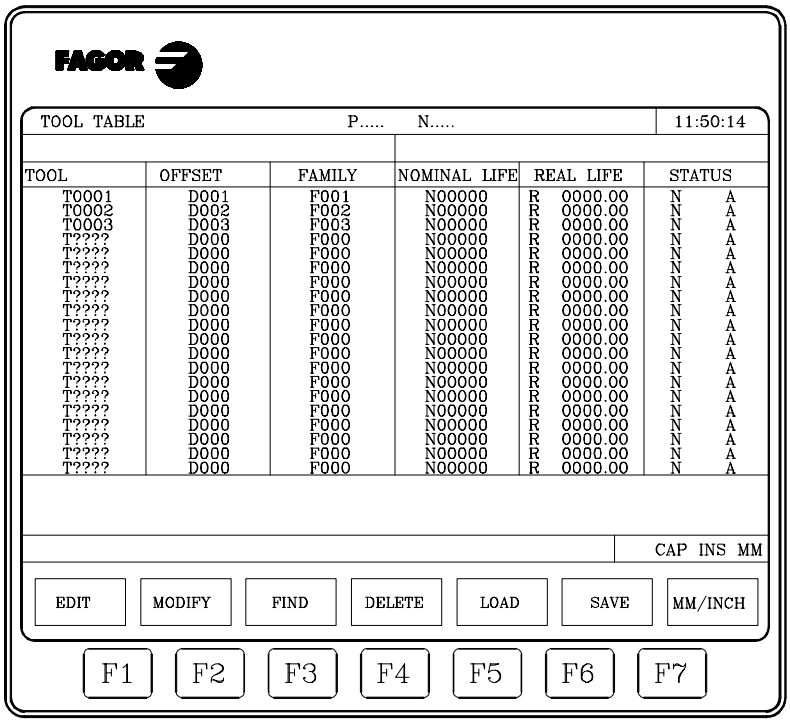
N = Normal (family 0-199) S = Special (family 200-255)

The tool status is defined as following:

A = Available
E = Expired (“real life” greater than “nominal life”)
R = Rejected by the PLC

6.3 TOOL TABLE

This table stores information about the tools available indicating the type of tool offset associated with them, their family, etc.



The end of this chapter describes how to edit the table. Each tool has the following data fields:

Offset number associated with the tool.

Every time a tool is selected, the CNC will assume its dimensions as they appear in the tool offset table for the specified tool offset.

Family code.

It will be used when having an automatic tool changer and it will allow replacing the worn out tool with another one with similar characteristics.

There are two types of families:

* Those for normal tools whose codes are between 0 and 199.

* Those for special tools (which occupy more than one magazine pocket), whose numbers are between 200 and 255.

Every time a new tool is selected, the CNC checks whether it is worn out ("real life" greater than "nominal life"). If that is the case, it will not select it, but it will select another one of the same family, instead.

If while machining a part, the PLC "asks" the CNC to reject the current tool (by activating the logic input “TREJECT”), the CNC will display the message "rejected" in the "STATUS" field and it will replace it with the next tool of the same family that appear in the tool table. This change will take place the next time that tool is selected.

Nominal tool life.

It indicates the machining time (in minutes) or the number of operations that that tool is calculated to last.

Real tool life.

It indicates the machining time (in minutes) or the number of operations already carried out by that tool.

Tool status.

It indicates the size of the tool and its status:

The tool size depends on the number of pockets it takes in the magazine and it is defined as follows:

N = Normal (family 0-199)
S = Special (family 200-255)

The tool status is defined as follows:

A = Available
E = Expired ("real life" greater than "nominal life")
R = Rejected by the PLC

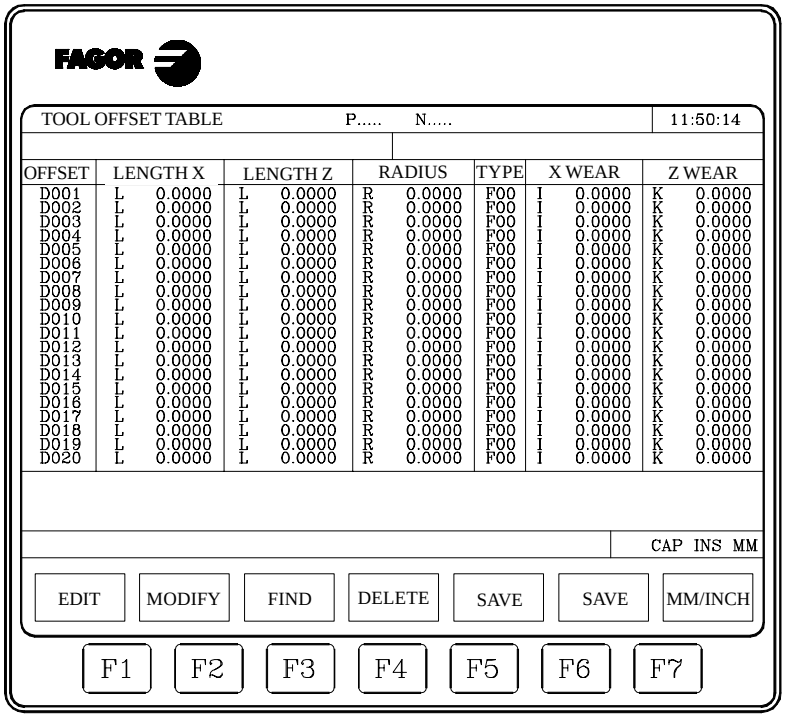
GEOMETRY

This softkey is available when the machine manufacturer has associated the tool geometry with the tool.

When selecting this option, the CNC lets access the tool geometry table. See section 6.5.

6.4 TOOL OFFSET TABLE

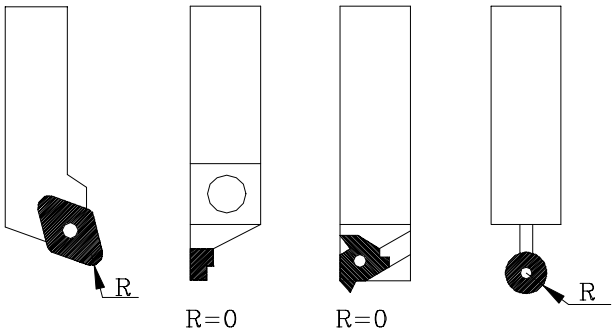
This table stores the dimensions of each tool.



The end of this chapter describes how to edit this table. Each tool offset has several fields defining the tool dimensions. These fields are:

Tool length along the X and Z axes. In radius

Tool radius.



Tool length wear (I) along the X axis.

It is given in diameters. The CNC will add this value to the nominal length along the X axis in order to calculate its real length (X+I).

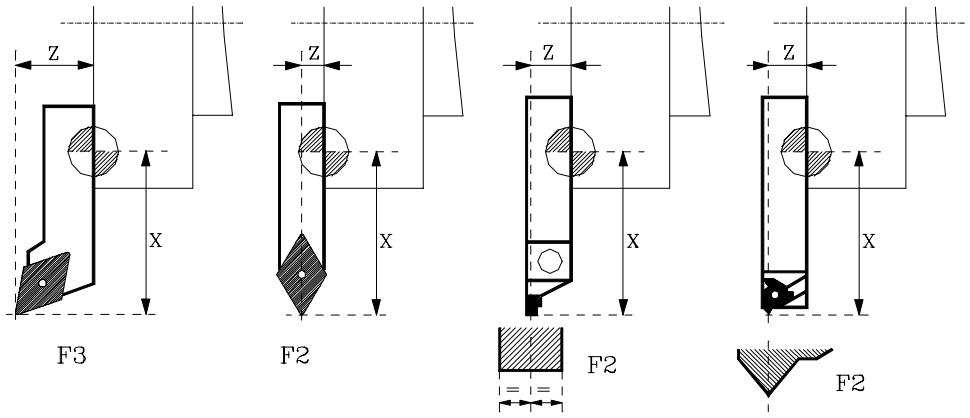
Tool length wear (K) along the Z axis.

It is given in diameters and in the work units indicated by the general machine parameter "INCHES". Its format is K ±5.5.

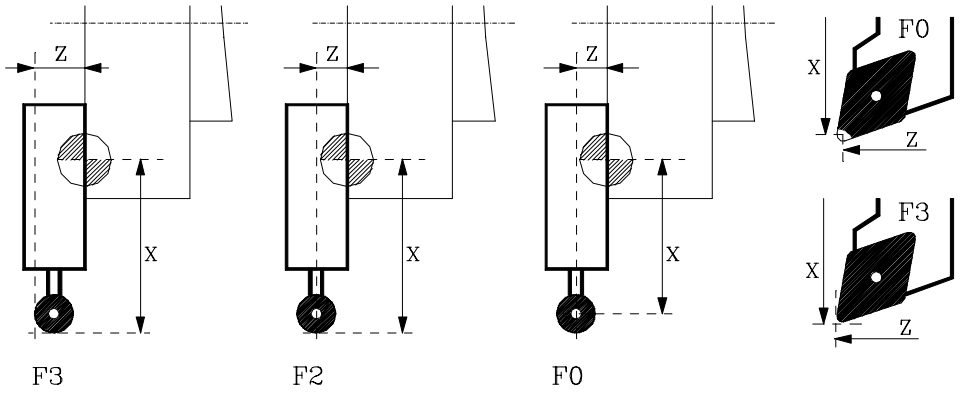
The CNC will add this value to the nominal length along the Z axis in order to calculate its real length (Z+I).

Tool type. location codes (shape) F0 through F10.

They indicate the tool shape and how it has been calibrated.



Codes F0 and F9 must only be used when calibrating the tool tip center and not the sides of the cutter.

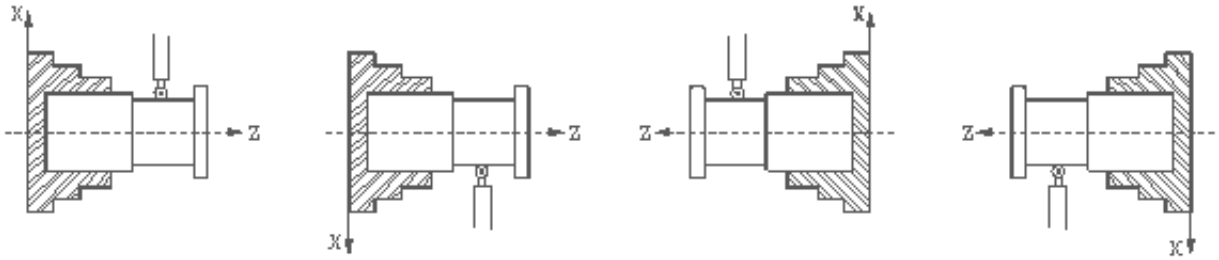


When using an end-mill or a drill bit, this location code must be F10.

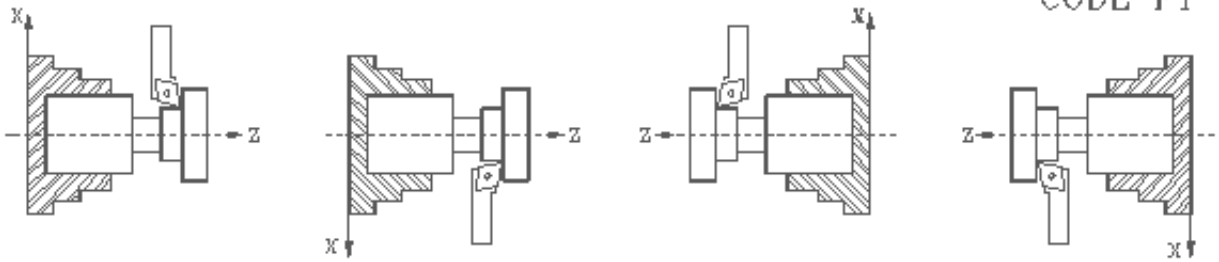
How to edit these values is described later on. They can also be modified from the PLC and from the part-program using the high level variables associated with the tools.

CODIGOS DE FORMA LOCATION CODES

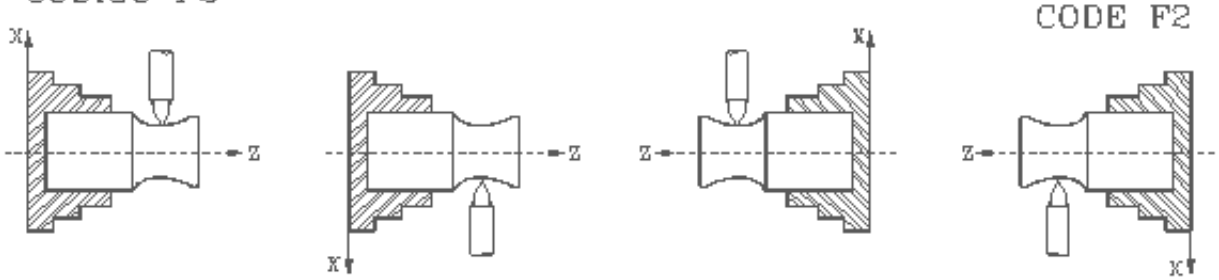
CODIGOS F0 y F9 CODES F0 and F9



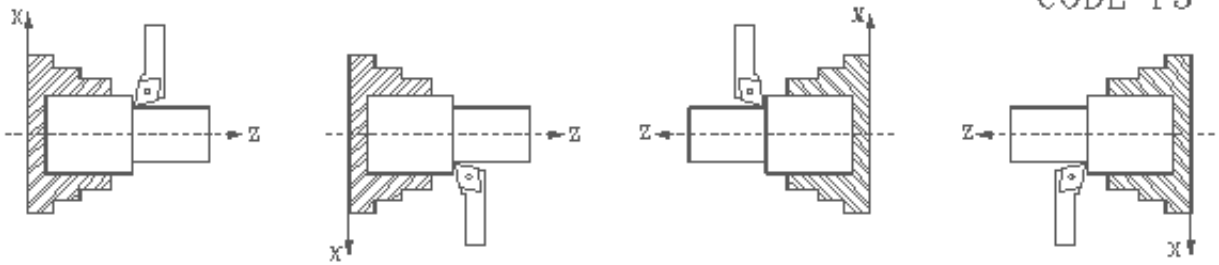
CODIGO F1 CODE F1



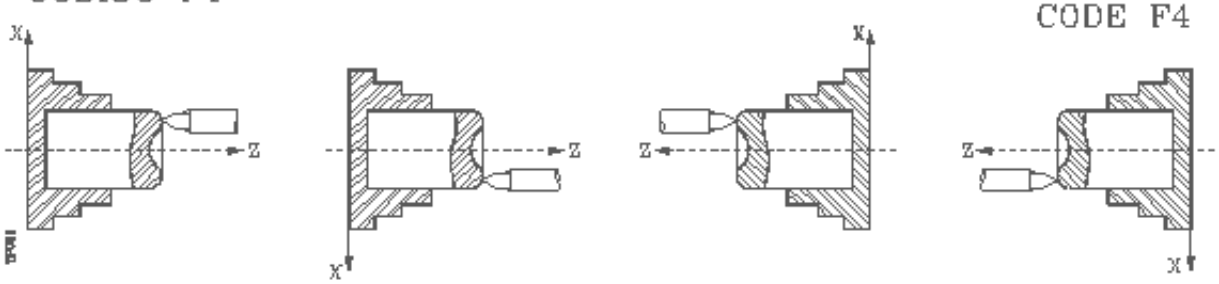
CODIGO F2 CODE F2



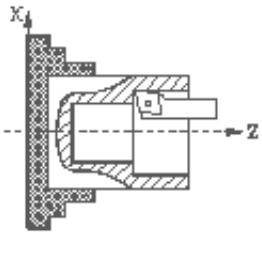
CODIGO F3 CODE F3

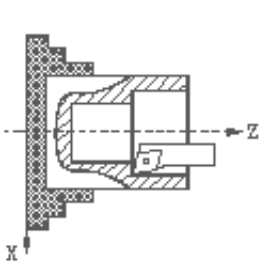


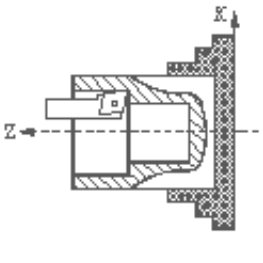
CODIGO F4 CODE F4



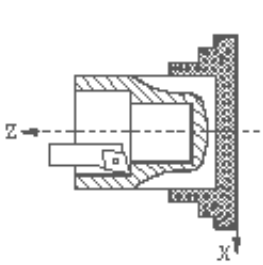
CODIGO F5



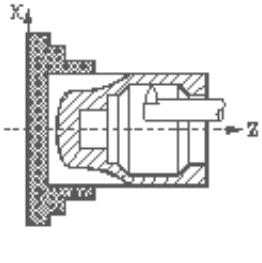


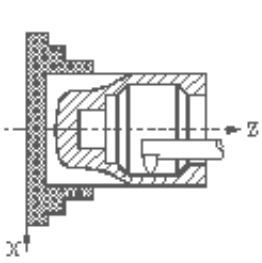


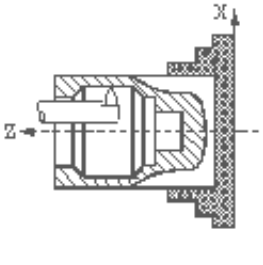
CODE F5



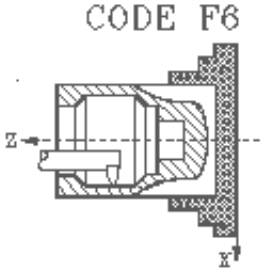
CODIGO F6



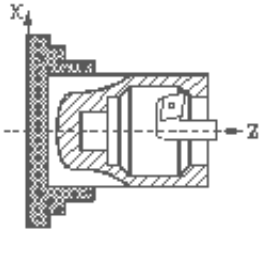


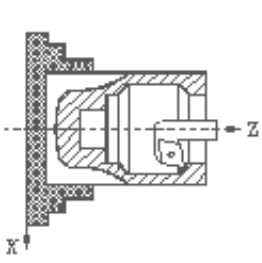


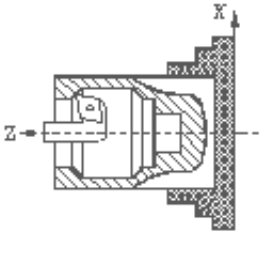
CODE F6



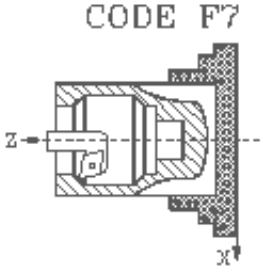
CODIGO F7



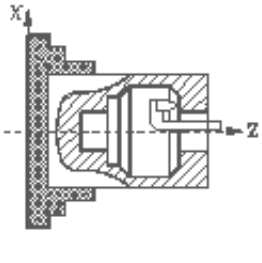


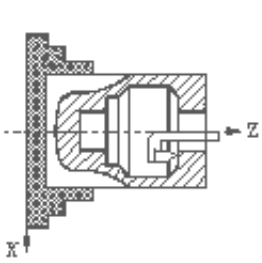


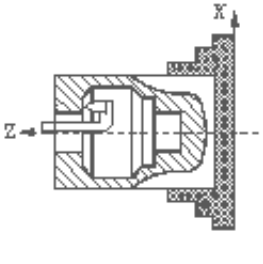
CODE F7



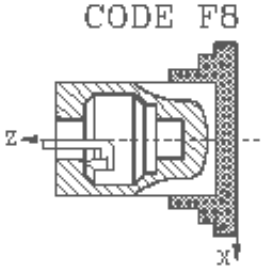
CODIGO F8



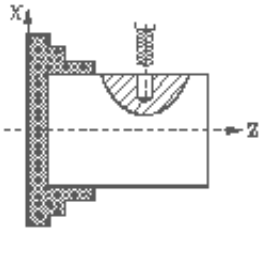


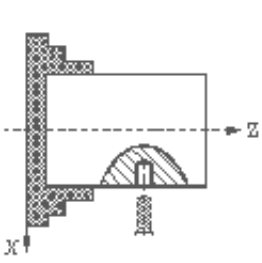


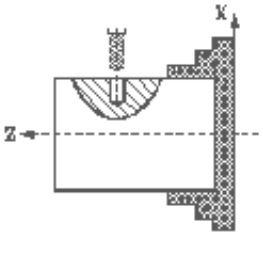
CODE F8



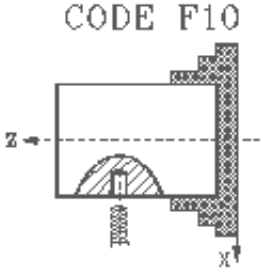
CODIGO F10



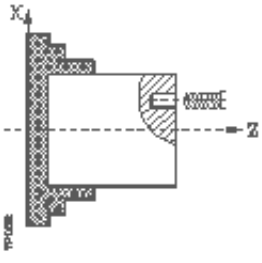


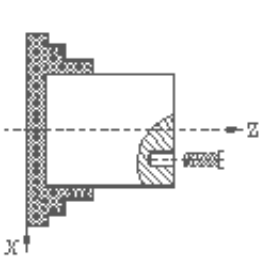


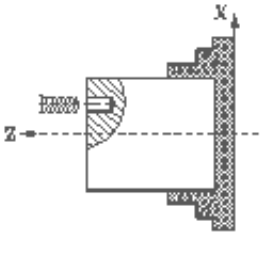
CODE F10



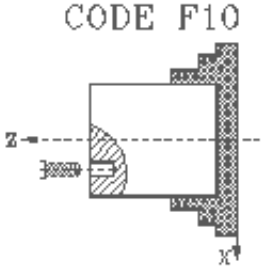
CODIGO F10







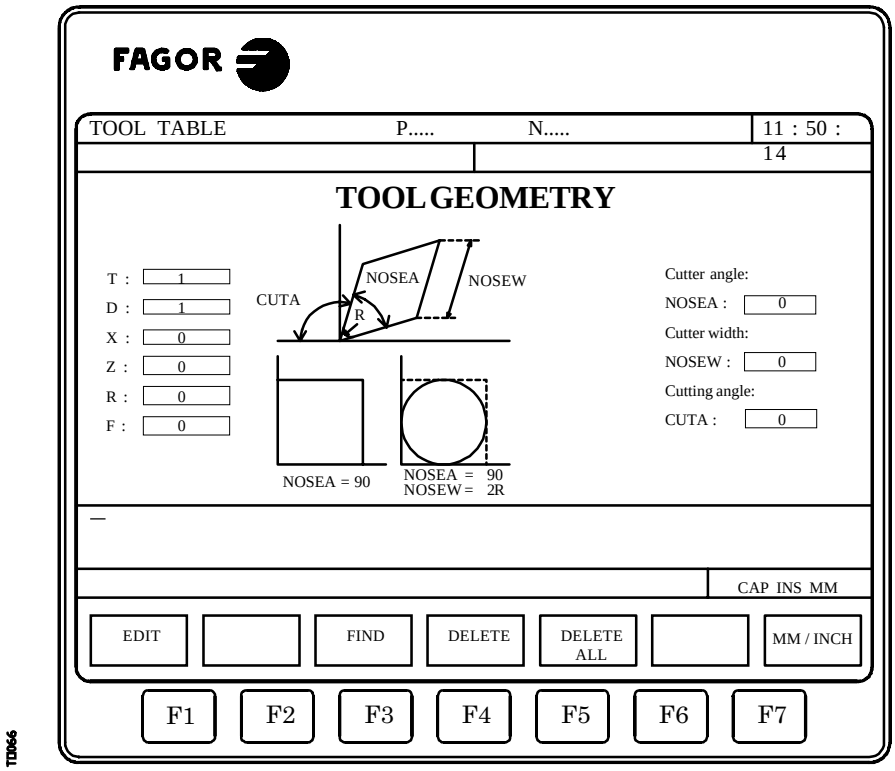
CODE F10



6.5 TOOL GEOMETRY TABLE

This table contains information such as cutter angle, cutting angles, etc. regarding the geometry of the available tools.

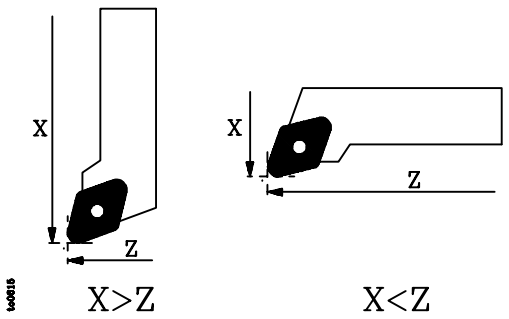
The data corresponding to each tool is contained in the following fields:



Tool data contained in other tool tables:

"T" Tool number
"D" Associated tool offset number (tool table)
"X" Tool length along X axis (tool offset table)
"Z" Tool length along Z axis (tool offset table)
"R" Radius of the cutter (tool offset table)
"F" Location code or type of tool (tool offset table)

If the "X" length assigned to a tool is greater than the "Z" length value, the CNC will show a tool with a vertical handle and when "X" is shorter than "Z", with a horizontal handle.



Cutter angle (NOSEA).

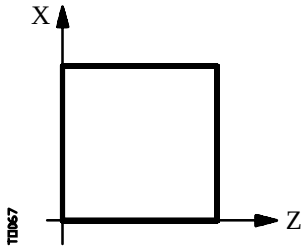
Indicates the angle formed by the sides of the cutter and it is given in degrees.

Cutter width (NOSEW).

Cutting angle (CUTA).

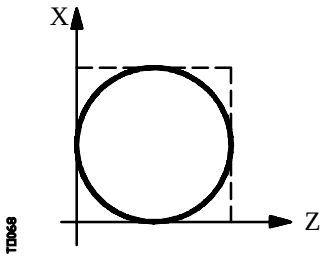
The parameters corresponding to the cutter angle (NOSEA) and the cutter width (NOSEW) must **always** be defined.

When the cutter is square, its angle value will be 90º (NOSEA = 90).



NOSEA=90

When the cutter is round, the cutter angle will be 90º (NOSEA=90) and the parameter corresponding to the cutter width must be assigned its diameter value (NOSEW = 2R).



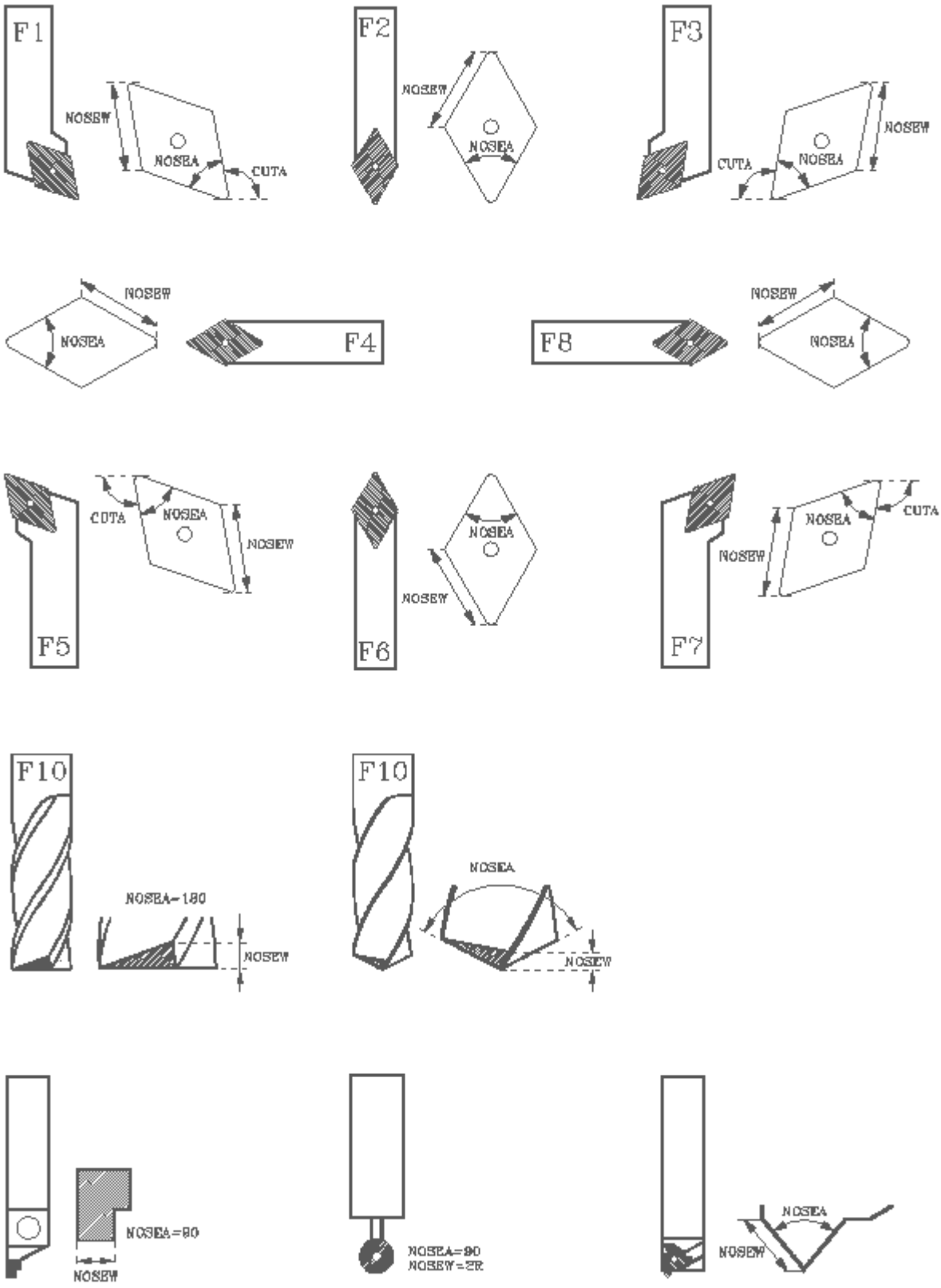
NOSEA=90
NOSEW=2R

The cutting angle (CUTA) must be defined [besides (NOSEA) and (NOSEW)] **only** when the type (location code) of the selected tool is F1, F3, F5 or F7.

If the selected tool type is F0 or F9, it is not necessary to define any parameter and if it is F1, F3, F5 or F7, the following parameters must be defined: the cutter angle (NOSEA), the cutter width (NOSEW) and the cutting angle (CUTA).

When the selected tool type is F2, F4, F6 or F8, only the cutter angle (NOSEA) and width (NOSEW) must be defined.

When using a drill bit or end-mill, Tool type (location code) F10 must be used being necessary to define the values of of NOSEW (cutter width) and NOSEA (cutter tip angle).


F1
NOSEW
NOSEA
CUTA
F2
NOSEW
NOSEA
F3
CUTA
NOSEA
NOSEW
NOSEW
NOSEA
F4
F8
NOSEW
NOSEA
F5
CUTA
NOSEA
NOSEW
F6
NOSEA
NOSEW
F7
NOSEA
CUTA
NOSEW
F10
NOSEA=180
NOSEW
F10
NOSEA
NOSEW
NOSEA=80
NOSEW
NOSEA=90
NOSEW=2R
NOSEA
NOSEW

6.5.1 HOW TO EDIT THE TOOL GEOMETRY TABLE

Once the tool geometry table is selected, the following options are available:

EDIT

Once this option is selected, the softkeys will change their background color to white and they will offer the following options:

- Cutter angle
- Cutter width
- Cutting angle

Previous menu. To return to the to the tool geometry menu.
Press **ESC** to quit this editing mode and return to the tool table menu.

FIND

With this function, it is possible to select a new geometry table. The CNC will request the number of the tool or offset associated with the geometry is to be displayed.

DELETE

With this option, it is possible to delete the geometry of the displayed tool setting to 0 its NOSEA, NOSEW and CUTA field values.

CLEAR ALL

With option it is possible to delete (set to 0) all the geometry fields (NOSEA, NOSEW and CUTA) of all the tables in the tool table.

MM / INCH

Every time this softkey is pressed, the CNC will change the display units for the coordinates corresponding to the tool geometry (X, Z, R). The units currently selected (mm or inches) will be displayed in the lower right-hand window.

6.6 GLOBAL AND LOCAL PARAMETER TABLES

The CNC has two types of general purpose variables:

Local parameters P0-P25 (7 levels)
Global parameters P100-P299.

The CNC updates the parameter tables after carrying out the operations indicated in the block in preparation. This operation is always carried out before executing the block. Therefore, the values shown in the table may not coincide with those of the block being executed.

When quitting the Execution mode after interrupting program execution, the CNC updates the parameter tables with the values corresponding to the block that was being executed.

In the global and local parameter tables, the values of the parameters may be displayed in decimal (4127.423) or in scientific notation (0.23476 E-3).

The CNC generates a new nesting level of local parameters every time parameters are assigned to a subroutine. Up to a maximum of 6 nesting levels of local parameters are possible.

Machining canned cycles G66, G68, G69, G81, G82, G83, G84, G85, G86, G87, G88 and G89 use the sixth nesting level of local parameters when they are active.

To access the different local parameter tables, the corresponding level must be indicated (0 through 6).

While programming in high level, local parameters may be referred to as P0-P25, or as **A-Z**,"A" being the same as "P0" and "Z" the same as "P25".

To do that, the local parameter tables show the letter associated to them, in brackets, next to the parameter number. In the tables, the parameter can only referred to as P0-P25, letters are not allowed.

6.7 HOW TO EDIT TABLES

The screen may be scrolled up and down line by line using the up/down arrow keys or page by page using the page up/down keys.

There are several ways to edit or modify a line which will be described next.

Once the user has selected any of those options, a editing area is available on the screen which may be scrolled up and down using the up/down arrow keys. On the other hand, with the up arrow key, the cursor may be placed over the first character of the editing window and, with the down arrow key over the last character.

EDIT

Once this option has been selected, the softkeys change color appearing over a white background and they show the information corresponding to the type of editing that may be done.

On the other hand, more information on the editing commands can be obtained at any time by pressing [HELP]. To quit this help mode, press [HELP] again.

Press [ESC] to quit the editing mode and maintain the table with the previous values.

Once the editing is done, press [ENTER]. The values assigned will be entered into the table.

MODIFY

Once this option has been selected, the softkeys change color appearing over a white background and showing the information corresponding to each field.

On the other hand, more information on the editing commands can be obtained at any time by pressing [HELP]. To quit this help mode, press [HELP] again.

By pressing [ESC], the information shown in the editing area is deleted. From this point on, the selected line may be edited again.

To quit the "modify" option, the information appearing in the editing area must be deleted by pressing [CL] or [ESC] and then [ESC]. The table will keep its previous values.

Once the modification is over, press [ENTER]. The new values assigned will be entered into the table.

FIND

Once this option has been selected, the softkeys will show the following options:

BEGINNING When pressing this softkey, the cursor is placed over the first line of the table that can be edited.

END When pressing this softkey, the cursor is placed over the last line of the table.

ZERO OFFSET, TOOL OFFSET, TOOL, POSITION, PARAMETER
When pressing one of these softkeys, the CNC requests the field number to be found. Once the field has been defined, press [ENTER].

The CNC searches for the requested field and places the cursor over it (when found).

DELETE

When deleting a line, the CNC sets all the fields to "0".

To delete a line, indicate its number and press [ENTER]

To delete several lines, indicate the beginning, press the [UP TO] softkey, indicate the last line to be deleted and press [ENTER]

To delete all the lines, press the "ALL" softkey. The CNC requests confirmation of the command.

INITIALIZE

It deletes all the data of the table by setting them all to "0". The CNC requests confirmation of the command.

LOAD

The tables may be loaded from the "Memkey Card" (CARD A) or a peripheral device or a PC through the two serial communications lines (RS232C or RS422).

The transmission starts after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the "ABORT" softkey.

If the length of the table received does not coincide with the current table length, the CNC will act as follows:

If the table received is shorter than the current one, the received lines are modified and the rest remain with their previous values.

If the table received is longer than the current one, all the lines of the table are modified and when detecting that there is no more room, the CNC will issue the corresponding error message

SAVE

The tables may be saved into the "Memkey Card" (CARD A) or out to a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission starts after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the "ABORT" softkey.

MM/INCHES

It toggles the display units for the data. The lower right-hand side window shows the units selected (MM/INCH).

7. UTILITIES

In this operating mode, one can access the programs stored in the CNC's RAM memory, in the "Memkey Card" (CARD A), in the hard disk (HD) and in external devices (through the serial lines 1 and 2).

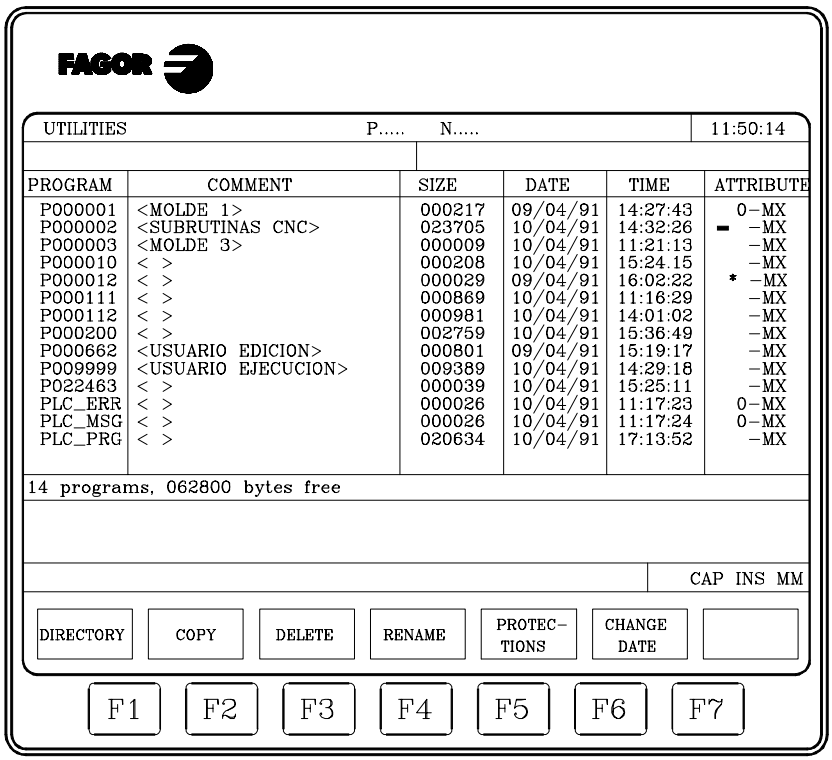
They can be deleted, renamed or their protection changed. It is also possible to make copies within the same device or from one to another.

7.1 DIRECTORY

To access the program directory of the CNC's RAM memory, the "Memkey Card" (CARD A), the hard disk (HD) and of the external devices (through serial lines 1 and 2).

The subroutine directory of the CNC can also be accessed.

Program directory.
By default, the CNC shows the program directory of the RAM memory, to view another directory, press the corresponding softkey.



On each directory, the CNC shows all the programs visible (not hidden) to the user, that is:

Part programs
Customizing programs
The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The program directory has the following definition fields:

Program
It shows the number when it is a part-program or a customizing program and the corresponding mnemonic when it is a PLC program, the PLC error file or the PLC message file.

Comment
Any program may have a comment associated with it for its identification.
The comments may be defined when editing the program or in this operating mode using the Rename option as described later on.

Size
It indicates, in bytes, the size of the program text. It must be borne in mind that the actual size of the program is slightly greater because this field does not include the space occupied by some variables used internally (header, etc.).

The date and the time when the program was edited (last changed)

Attributes
They show information about the source and usefulness of each program. The attributes are defined in this operating mode by means of the Protections option as described later on.

***** The program is running, either because it is the main program or because it contains a subroutine which has been called upon from that program or from another subroutine.

O The program was created by the machine manufacturer.

H The program is hidden and cannot be displayed in any directory.

Since a hidden program can nevertheless be edited or deleted if its number is known, it is recommended to remove the "Modifiable" attribute to prevent it from being edited or deleted.

M The program may be modified. In other words, it may be edited, copied, etc.

If a program does not have this attribute, the operator cannot see or modify its contents.

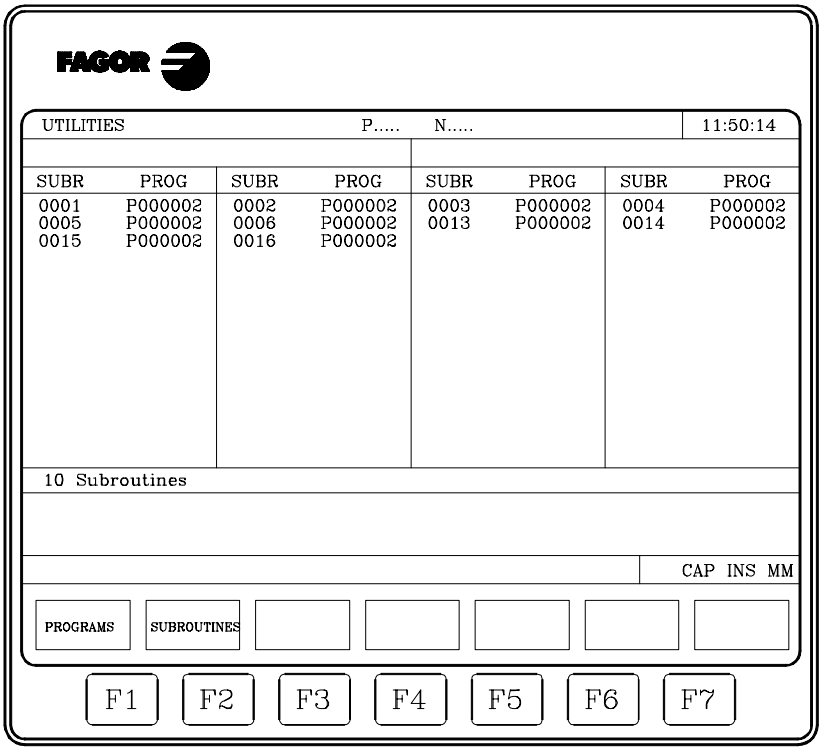
X Indicates that the program may be executed.

A program not having this attribute cannot be executed by the operator.

Only the attributes currently selected will be shown, the ones not selected will appear as "-".

Example: O—X Indicates that the program was created by the manufacturer, it will be displayed in the directory (not hidden), it cannot be modified, but it may be executed.

Subroutines directory.



It lists all the subroutines defined in the part programs of the CNC ordered from the smallest one to the largest one.

Also, next to the subroutines, it displays the number of the program where it has been defined.

If the program containing the subroutine has the "hidden" attribute assigned to it, that program number will appear as P??????.

7.1.1 DIRECTORY OF THE EXTERNAL DEVICES

When accessing the directory of an external device through the serial lines, that directory is shown in DOS format.

The [CHANGE DIR] softkey lets the user select the work directory of the PC to operate with from the CNC.

This operation does not change the work directory that was selected to operate with from the PC.

In other words, when working via DNC, it is possible to select a work directory at the PC and another PC directory at the CNC.

This new feature is available from DNC50 version 5.1 on.

7.2 COPY

To copy programs in the same directory or between directories of different devices.

The copies may be made between:
The CNC's RAM memory, "Memkey Card" (CARD A), hard disk (HD) and external devices (serial lines)

To make a copy, proceed as follows:

Press the [COPY] softkey
Indicate the location of the program to be copied (RAM memory, CARD A, HD or DNC)
key in the program number to be copied
Press the [IN] softkey
Indicate the destination of the copy (RAM memory, CARD A, HD or DNC)
Key in the program number
Press [ENTER]

Example to copy program 200103, from the CNC's RAM memory out to the "Memkey Card" with the number 14

COPY (MEMORY) P200103 IN (CARD A) P14 ENTER

If a program with the same number already exists, the CNC will display a warning message. On the other hand, if that program is in execution, the CNC will display a message indicating that it is not possible.

Two subroutines may not have the same name in RAM memory. To make a copy and change the name of the copied subroutine, write the subroutine defining block as a comment **before** making the copy.

7.3 DELETE

A program may be deleted from the CNC' RAM memory, from the "Memkey Card" (CARD A), from the Hard Disk (HD) or from the external devices (through the serial lines 1 and 2).

To delete a program, proceed as follows:

Press the [DELETE] softkey
Indicate the location of the program to be deleted (RAM, CARD A, HD or DNC).
Key in the number of the program to be deleted or place the cursor over it.
Press [ENTER]

Example to delete program 200103 from the "Memkey Card"

DELETE (CARD A) P200103 ENTER

Only programs that can be modified ("M" attribute) can be deleted.

7.4 RENAME

To rename or assign a new comment to a program stored in the CNC's RAM memory, "Memkey Card" (CARD A), or in the Hard Disk (HD).

To rename a program, proceed as follows:

Press the [RENAME] softkey.
Indicate the location of the program (RAM, CARD A, HD or DNC).
Key in the number of the program to be renamed
Press the [TO] softkey.
Press either the [NEW NUMBER] or the [NEW COMMENT] softkey
Key in the new number or the new comment
Press [ENTER].

The files associated with the PLC (program, messages and errors) are always referred to with their associated mnemonics. Therefore, only their comment may be renamed.

If there is a program with the same number, the CNC will issue a warning message and it will offer the chance to modify the command.

Examples:

To change the name of program 200103 from the "Memkey Card"

RENAME (CARD A) P200103 TO NEW NUMBER P12 ENTER

to change the comment of program 100453 from the CNC

RENAME (MEMORY) P100453 TO NEW COMMENT "Test" ENTER

7.5 PROTECTIONS

To prevent certain programs from being manipulated and restrict access to the operator to certain CNC commands.

It is possible to protect programs stored in the CNC's RAM memory, in the "Memkey card" (CARD A) or in the Hard Disk (HD).

USER PERMISSIONS

Lets the operator see those CNC programs that have been created by the operator and sets their attributes.

To modify the attributes of a program, proceed as follows:
Press the [USER PERMISSION] softkey
Indicate the location of the program (RAM MEMORY, CARD A or HD)
Key in the number of the program whose attribute is to be changed
Press the softkeys

F2	to change the (H) attribute	hidden/visible program
F3	to change the (M) attribute	modifiable program
F4	to change the (X) attribute	executable program

Press [ENTER]

OEM PERMISSION

Lets see all the programs stored at the CNC whether they are created by the OEM or by the operator and set their attributes.

To modify the attributes of a program, proceed as follows:
Press the [OEM PERMISSION] softkey
Indicate the location of the program (RAM MEMORY, CARD A or HD)
Key in the number of the program whose attribute is to be changed
Press the softkeys

F1	to change the (O) attribute	OEM program
F2	to change the (H) attribute	hidden/visible program
F3	to change the (M) attribute	modifiable program
F4	to change the (X) attribute	executable program

Press [ENTER]

PASSWORDS

Lets define each of the passwords that the operator must key in before accessing the various CNC commands.

General access password (MASTERPSW)
It is requested when trying to access this password option.
(Utilities mode / Protections / Passwords).

OEM password (OEMPSW)
It is requested when trying to access OEM permissions
(Utilities mode / Protections / OEM permissions).

User password (USERPSW)
It is requested when trying to access user permissions
(Utilities mode / Protections / User permissions).

PLC access password (PLCPSW)

It is requested in the following cases:

- When editing the PLC program, the PLC message program and the PLC error program.
- When compiling the PLC program.
- When trying to change the status of a resource or execute a program execution controlling command.

Customizing password (CUSTOMPSW)

It is requested when trying to access the Customizing mode or when attempting to erase an OEM screen.

Machine parameter access password (SETUPPSW)

It is requested when trying to access the options to modify the table values (Edit, Modify, Initialize, Delete and Load) except for tables of the serial lines which are not protected.

To change or delete the passwords, use the following softkeys:

Change password.

Select the desired password and enter the new one.

Delete password.

Lets delete (eliminate) one of several codes from the table.

- To delete a password, indicate its number and press [ENTER].
- To delete several passwords (they must be in a row), indicate the number of the first one to be delete, press the "UPTO" softkey, indicate the number of the last one to be deleted and press [ENTER].
- To delete a password, indicate its number and press [ENTER].

Clear all.

Lets delete all the passwords. The CNC will request confirmation of the command and it will delete them after pressing [ENTER].

7.6 CHANGE DATE

Lets change the system date and time.

First, the date will be shown as day/month/year (12/04/1998). After changing it, press [ENTER] to validate it. If it is not to be changed, press [ESC].

Next, the time will be shown as hours/minutes/seconds (08/30/00). After changing it, press [ENTER] to validate it. If it is not to be changed, press [ESC].

8. DNC

This operating mode shows the CNC status after a power outage and the status of the DNC communication lines. It also lets activate and deactivate DNC communications with a PC.

8.1 CNC

This screen shows the number of the line that was being executed last time an execution error or a power outage occurred.

The CNC shows the program number and line number that was executing as well as where the program is stored.

On the other hand, if that program called upon a subroutine and the CNC was executing it, it will display:

The subroutine number, the program containing its definition and the line or block of the subroutine being executed.

Example:

Device	Program	Line number	Subroutine
CARD A	000012	7	
MEMORY	001000	15	0033

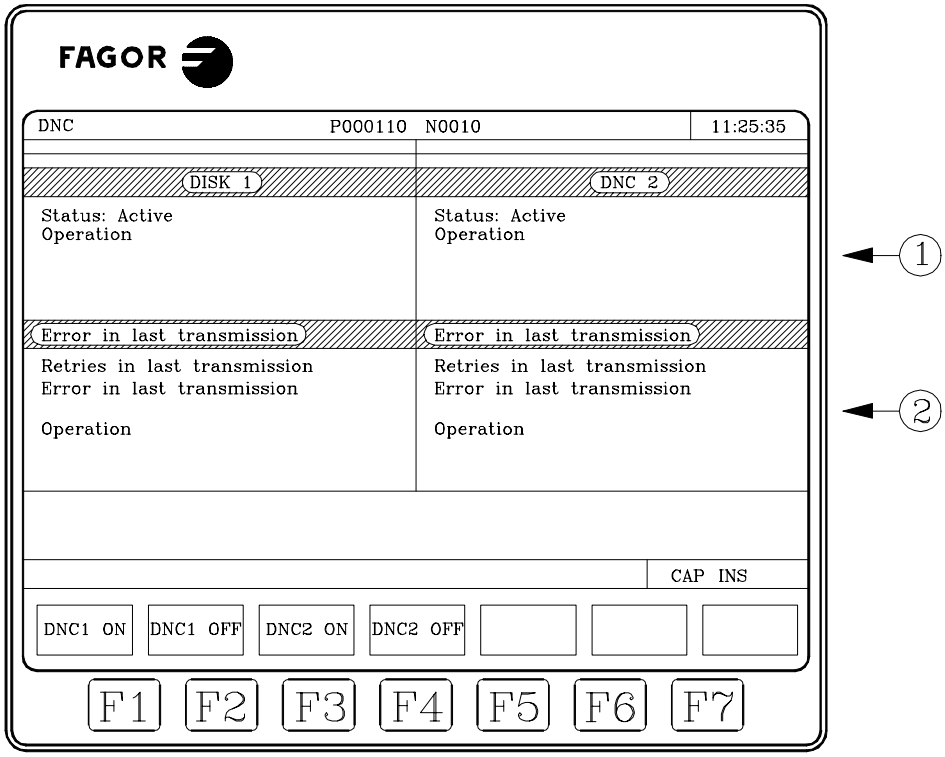
Indicates that the CNC was executing line 7 of program 12 of CARD A.

That program line called to subroutine 15 and it was executing its line number 33. That subroutine is defined (contained) in program 1000 which is stored in the CNC's RAM memory.

8.2 DNC

With this CNC, it is possible to access this operating mode when at least one of the serial lines (RS232C or RS422) is set to work in the DNC mode or to communicate with the FAGOR Floppy Disk Unit.

When accessing this mode, the CNC shows the following screen:



The left-hand side of the screen corresponds to serial line 1 and the right-hand side to serial line 2.

In the example of the figure above, serial line 1 is used to communicate with a Fagor Floppy Disk Unit; and serial line 2 to communicate via DNC.

The upper area, 1, indicates:

* The status of the serial line: Active / Inactive.

* The type of operation in progress:
  Sending program / Receiving program / Sending directory / Receiving directory / etc.

The lower area, 2, indicates the last operation and the type of error occurred if any.

9. *PLC*

In this mode of operation it is possible to access the PLC to check its operation or the status of the various PLC variables. It also allows editing and analyzing the PLC program as well as the PLC message file and error file.

The accessible programs associated with the PLC are:

The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The PLC program (PLC_PRG) may be edited at the front panel or copied from the "Memkey Card" (CARD A) or from a peripheral device or PC.

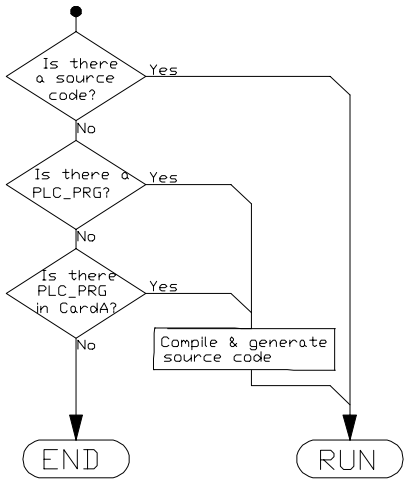
The PLC program (PLC_PRG) is stored in the internal CNC memory with the part-programs and it is displayed in the program directory (utilities) together with the part-programs.

Before executing the PLC_PRG program, it must be compiled.
Once it is done compiling, the CNC requests whether the PLC should be started or not.

To make the operator life easier and avoid new compilations, the source code generated at each compilation is stored in memory.

After power-up, the CNC acts as follows:

- Runs the executable program stored in memory.
- If there isn't one, it compiles the PLC_PRG program already in memory and runs the resulting executable program.
- If there isn't one, it looks for it in the "Memkey Card" (CARD A)
- If it isn't in the CARD A either, it does nothing. Later on, when accessing the Jog mode, Execution mode, etc. the CNC will issue the corresponding error message.



Once the program has been compiled, it is not necessary to keep the source program (PLC_PRG) in memory because the PLC always executes the executable program.

Once the proper performance of the PLC has been verified, it is a good idea to save it into the "Memkey Card" (CARD A) using the instruction SAVE PROGRAM (as described later on).

9.1 EDIT

Once this option is selected, indicate with the corresponding softkey the PLC program to be edited.

The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The cursor can be moved line by line with the “up and down” arrow keys or page by page with the “page up” and “page down” keys.

The cursor position or line number will be displayed in a white window inside the communications window (bottom of the screen) next to the CAP/INS indicator window.

This operating mode offers various options which are described next.

Once any of these functions is selected, the CNC shows an editing area on the CRT where the cursor may be moved by using the up/down and right/left arrow keys. Also, the up-arrow key positions the cursor over the first character of the editing area and the down-arrow key positions the cursor over the last character.

EDIT

With this option it is possible to edit new lines or blocks of the selected program.

Before pressing this softkey, the block after which the new ones will be added must be selected with the cursor.

The program will be edited (written) a block at a time and each block can be written in ISO language, High Level language or it can be just a program comment.

Once this option is selected, the softkeys will change their color showing their type of editing option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

Press the **ESC** key to exit the block editing mode when writing a block and this block will not be added to the program.

Once the block has been edited, press **ENTER** to add it to the program behind the block previously indicated by the cursor.

The cursor will be positioned at the new block (just edited) and the editing window (area) will be cleared In order to edit a new block.

Press **ESC** or **MAIN MENU** to quit the block editing mode.

MODIFY

This option permits modifying the contents of a selected program block.

Before pressing this softkey, select with the cursor the block to be modified .

Once this option is selected, the softkeys will change their color showing their type of modifying option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

By pressing **ESC**, the information corresponding to that block and which was shown in the editing area will be cleared. It will then be possible to modify its contents again.

To quit the block modifying mode, press **CL** or **ESC** to clear the editing window and then press **ESC** again. This way, the selected block will not be modified.

Once the block contents have been modified, press **ENTER** so the new contents replace the old ones.

FIND

This option is used to find a specific text within the selected program.

When selecting this option, the following options will appear:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

When the text is defined, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode. The cursor will be positioned where the indicated text was found last.

LINE NUMBER After pressing this key, the CNC requests the number of the block to be found. After keying in the desired number and pressing **ENTER**, the cursor will position over that block which will then be selected quitting the search mode.

REPLACE

With this function it is possible to replace a character sequence with another throughout the selected program.

When selecting this option, the CNC requests the character sequence to be replaced.

Once the text to be replaced is indicated, press the "WITH" softkey and the CNC will request the character sequence which will replace the previous one.

Once this text is keyed in, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of the searched text.

The search will begin at the current block.

The found text will be highlighted and the following softkey options will appear:

REPLACE Will replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

DO NOT REPLACE Will not replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

TO THE END This function will automatically replace all the matching text from the current block to the end of the program without offering the option of not replacing it.

ABORT This function will not replace the highlighted text and it will quit the "find and replace" mode.

DELETE BLOCK

With this function it is possible to delete a block or group of blocks.

To delete only one block, just position the cursor over it and press **ENTER**.

To delete a group of blocks, indicate the first and last blocks to be deleted. To do so, follow these steps:

* Position the cursor over the first block to be deleted and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be deleted and press the "FINAL BLOCK" softkey.

  If the last block to be deleted is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to delete them.

MOVE BLOCK

With this option it is possible to move a block or group of blocks by previously indicating the first and last blocks to be moved. To do so, follow these steps:

* Position the cursor over the first block to be moved and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be moved and press the "FINAL BLOCK" softkey.

  If the last block to be moved is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To move only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to move them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the "START OPERATION" softkey to carry out the move.

COPY BLOCK

With this option it is possible to copy a block or group of blocks by previously indicating the first and last blocks to be copied. To do so, follow these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to copy them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the "START OPERATION" softkey to carry out this command.

COPY TO PROGRAM

With this option it is possible to copy a block or group of blocks of one program into another program.

When selecting this option, the CNC will request the number of the destination program where the selected block or blocks are to be copied. After entering the program number press **ENTER**.

Next, indicate the first and last blocks to copy by following these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks and will execute the command.

  If the destination program already exists, the following options will be displayed:

* Write over the existing program. All the blocks of the destination program will be erased and will be replaced by the copied blocks.

* Append (add) the copied blocks behind the ones existing at the destination program.

* Abort or cancel the command without copying the blocks.

INCLUDE PROGRAM

With this option it is possible to include or merge the contents of another program into the one currently selected.

Once this option is selected, the CNC will request the number of the source program to be merged. After keying in that number press **ENTER**.

Next, indicate with the cursor the block after which the source program will be included.

Finally, press the "START OPERATION" softkey to execute the command.

9.2 COMPILE

With this option it is possible to compile the PLC source program “PLC_PRG”.

The PLC program must be stopped in order to compile it, otherwise, the CNC will “ask” if it is desired to stop it.

Once the source program compiled, the CNC will generate the executable PLC program (object program).

If while compiling, some errors are detected, the CNC will not create the object program and the detected errors (up to 15) will appear on the screen.

If the errors do not affect the proper program execution (such as non-referenced labels, etc.), the CNC will display the corresponding warning messages but it will generate the object program.

After a successful compilation, the CNC will “ask” whether the PLC program must be started or not.

9.3 MONITORING

With this option it is possible to display the PLC program and analyze the status of the different PLC resources and variables.

Once this option has been selected, the CNC will show the source program that corresponds to the executable program (object) even when that program (source) has been deleted or modified at the CNC.

The CNC will also display all the variable consultations at logic level 1 (including those not being executed) and the actions whose conditions are met.

To display the program from a specific line on, press the "**L**" key followed by that line number and then press **ENTER**.

The operator can move the cursor around the CRT a line at a time with the up/down arrow keys and a page at a time with the page-up and page-down keys.

The various monitoring options available are described next.

Once any of the those options has been selected, the operator has an editing window where the cursor may be moved with the right and left arrow keys. The up arrow will position the cursor over the first character of the editing window and the down arrow over the last one.

MODIFY THE STATUS OF THE RESOURCES

The CNC has the following instructions to modify the status of the different PLC resources.

I 1/256 = 0/1	Alters the status (0/1) of the indicated input. For example: I120 = 0, sets input I120 to 0.
I 1/256.1/256 = 0/1	Alters the status (0/1) of a the indicated group of inputs. For example: I100.103 = 1, sets inputs I101, I102 and I103 to 1.
O 1/256 = 0/1	Alters the status (0/1) of the indicated output. For example: O20 = 0, sets output O20 to 0.
O 1/256.1/256 = 0/1	Alters the status (0/1) of the indicated group of outputs. For example: O22.25= 1 sets outputs O22 thru O25 to 1.
M 1/5957 = 0/1	Alters the status (0/1) of the indicated mark. For example: M330 = 0 sets Mark M330 to 0.
M 1/5957.1/5957 = 0/1	Alters the status (0/1) of the indicated group of marks. For example: M400.403=1 sets marks M400 thru M403 to 1.
TEN 1/256 = 0/1	Alters the status (0/1) of the ENABLE input of the indicated timer. For example: TEN12 = 1, sets the Enable input of timer T12 to 1.

TRS 1/256 = 0/1	Alters the status (0/1) of the RESET input of the indicated timer. For example: TRS2 = 0 sets the reset input of timer T2 to 0.
TGn 1/256 n = 0/1	Alters the status (0/1) of the trigger input "TGn" of the indicated timer (1 thru 256) assigning the desired time constant (n) to it. For example: TG1 22 1000 sets the trigger input 1 of timer T22 to one and it assigns a time constant of 1000 (10 seconds).
CUP 1/256 = 0/1	Alters the status (0/1) of the UP count input of the indicated counter. For example: CUP 33 = 0 sets the status of the UP input of counter C33 to 0.
CDW 1/256 = 0/1	Alters the status (0/1) of the DOWN count input of the indicated counter. For example: CDW 32 = 1 sets the status of the UP input of counter C32 to 1.
CEN 1/256 = 0/1	Alters the status (0/1) of the enable input of the indicated counter. For example: CEN 12 = 0, sets the enable input of counter 12 to 0.
CPR 1/256 n = 0/1	Alters the status (0/1) of the preset input of the indicated counter (1 thru 256). The counter will be preset with the value "n" if an up flank is produced with this instruction. For example: CPR 10 1000 =1 sets the preset input of counter C10 to 1 and also, if an up flank has occurred (being previously set to 0), the counter will be preset with a value of 1000.
C 1/256 = n	Presets the count of the indicated counter to the "n" value. For example: C42 = 1200 sets the count of counter C42 to 1200.
B 0/31 R 1/559 = 0/1	Alters the status (0/1) of the indicated bit (0/31) of the indicated register (1/559). For example: B5 R200 = 0 sets Bit 5 of register R200 to 0.
R 1/559 = n	Assigns the "n" value to the indicated register. For example: R 303 = 1200 assigns the value of 1200 to register R303.
R 1/559.1/559 = n	Assigns the "n" value to the indicated register group. For example: R234.236 = 120 assigns the value of 120 to registers R234, R235 and R236.

It must be borne in mind that when referring to a single resource, it is possible to do it using its corresponding mnemonic.

For example: /STOP=1 is interpreted by the CNC as M5001=1

CREATE WINDOW

This CNC allows the possibility of creating windows to display the status of the various PLC resources.

These windows will be shown overlapping the PLC program and the information displayed in them will be updated dynamically.

The options "MODIFY WINDOW", "ACTIVE WINDOW" and "ACTIVATE SYMBOLS" allow the manipulation of these windows.

Every time a new window is created, the CNC will assign 2 data lines to it in order to display the status of the desired resources.

There are two types of windows which can be selected with softkeys.

WINDOW TO DISPLAY TIMERS AND REGISTERS

This window is divided into two sections, one to display Timers and the other one to display Registers.

Timer. It will show one timer per line showing the following information for each one of them:

TG	Indicates the logic status of the active trigger input.
M	Indicates the status of the timer: "S" means stopped, "T" means timing and "D" means disabled.
TEN	Indicates the logic status of the Enable input.
TRS	Indicates the logic status of the Reset input.
T	Indicates the logic status of the status output of the timer.
ET	Indicates the elapsed time.
TO	Indicates the remaining time.

Key in the command **T 1/256** or **T 1/256.1/256** to request the data on a timer or group of timers and then press **ENTER**.

Register. It will display one register per line showing the following information fields for each of them:

HEX	Indicates the hexadecimal value of its contents.
DEC	Indicates the decimal value of its contents (with sign).

Key in **R 1/559** or **R 1/559.1/559** to request information on one or more registers and, then, press **ENTER**.

WINDOW TO DISPLAY COUNTERS AND BINARY DATA

This window is divided into two sections, one to display Counters and the other one to display Binary Data.

Counter. It will display one counter per line showing the following information fields for each of them:

CEN	Indicates the logic status of the ENABLE input.
CUP	Indicates the logic status of the UP COUNT input.
CDW	Indicates the logic status of the DOWN COUNT input.
CPR	Indicates the logic status of the PRESET input.
S	Indicates the status of the counter. "1" when its internal count is 0 and 0 for all other cases.
C	Indicates its count value.

Key in **C 1/256** or **C 1/256.1/256** to request information on one or more counter and, then, press **ENTER**.

Binary Data. It will show one data line per resource or group of resources requested.

The instructions available to request information of the various resources are:

I 1/256 or I 1/256.1/256	It shows the status of the selected input or group of inputs.
O 1/256 or O 1/256.1/256	It shows the status of the selected output or group of outputs.
M 1/5957 or M 1/5957.1/5957	It shows the status of the selected mark or group of marks.
B 0/31 R 1/559	It shows the status of the selected bit of the indicated register.

When requesting the status of one or more inputs, outputs or marks, the CNC will show complete data lines even when all of them have not been requested.

When using generic denominators (I / O / M) to display resources, the CNC will display 20 of them per line and 3 when using their associated mnemonics (symbols). In the latter case, the generic denomination will be displayed when no mnemonic is associated to a resource.

When requesting the status of a register bit, the CNC will display only the requested bit on the corresponding line.

MODIFY WINDOW

With this option it is possible to manipulate the active window (the one selected) by enlarging it, reducing it, clearing it or even eliminating (closing) it.

To do so, the following softkey options are available:

ENLARGE	To enlarge the size of the window by one line every time this softkey is pressed.
REDUCE	To reduce the size of the window by one line every time this softkey is pressed (minimum 2 lines).
CLEAR	To clear the contents of the active window.
CLOSE	To close the active window, the CNC will no longer display it.

ACTIVE WINDOW

With this option it is possible to select between the PLC program and each one of the windows being displayed (timers, registers, counters and binary data) in order to operate with it.

Bear in mind that the operator can only operate with the active window.

Once the active window has been selected, it Will be possible to:

Move the cursor (if the PLC program is the one active) or shift the display area with the up and down arrow keys.

Execute any command of the “MODIFY WINDOW” option.

FIND

This option will be executed regardless of which is the active window and it offers the following searching options:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

The CNC will consider a text found when it is isolated by blank spaces or separators. Thus, When looking for "I1" it will not find or stop at "I12" or "I123", but only at "I1".

When the text is defined, Press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode.

ACTIVATE / DEACTIVATE SYMBOLS

With this option it is possible to display in all available windows the symbols or mnemonics associated to the various resources.

The names of the resources may be displayed in two ways: using their generic names (I, O, M, T, C, R) by **deactivating** symbols or using their associated symbols by **activating** them.

When a resource has no mnemonic associated to it, it will always be displayed with its generic name.

This softkey will toggle between ACTIVATE SYMBOL and DEACTIVATE SYMBOL every time is pressed in order to show which option is available.

LOGIC ANALYZER

It is especially suited to help in the machine startup and for troubleshooting errors and critical situations in signal behavior.

START PLC

When selecting this option, the CNC will start executing the PLC program from the beginning, including the CY1 cycle.

The CNC will ignore this command when it is already executing the PLC program.

FIRST CYCLE

When selecting this option, the CNC will execute only the initial cycle of the PLC program (CY1).

The CNC will ignore this command when it is already executing the PLC program.

SINGLE CYCLE

When selecting this option, the CNC will execute the main cycle of the PLC program (PRG) only once.

The CNC will ignore this command when it is already executing the PLC program.

STOP PLC

This softkey interrupts the execution of the PLC program.

CONTINUE

This softkey resumes the execution of the PLC program.

9.3.1 MONITORING WITH THE PLC IN OPERATION AND WITH THE PLC STOPPED

It must be borne in mind that the CNC initializes all physical outputs and the PLC resources on power-up, after the key sequence SHIFT-RESET and after detecting a WATCHDOG error at the PLC.

The initialization process sets all resources to "0" except those active low. They will be set to "1".

During the monitoring of the PLC program and the various PLC resources, the CNC will always show the real values of the resources.

If the PLC is on, note that a program cycle is processed in the following way:

* The PLC updates the real input values after reading the physical inputs (from the electrical cabinet).

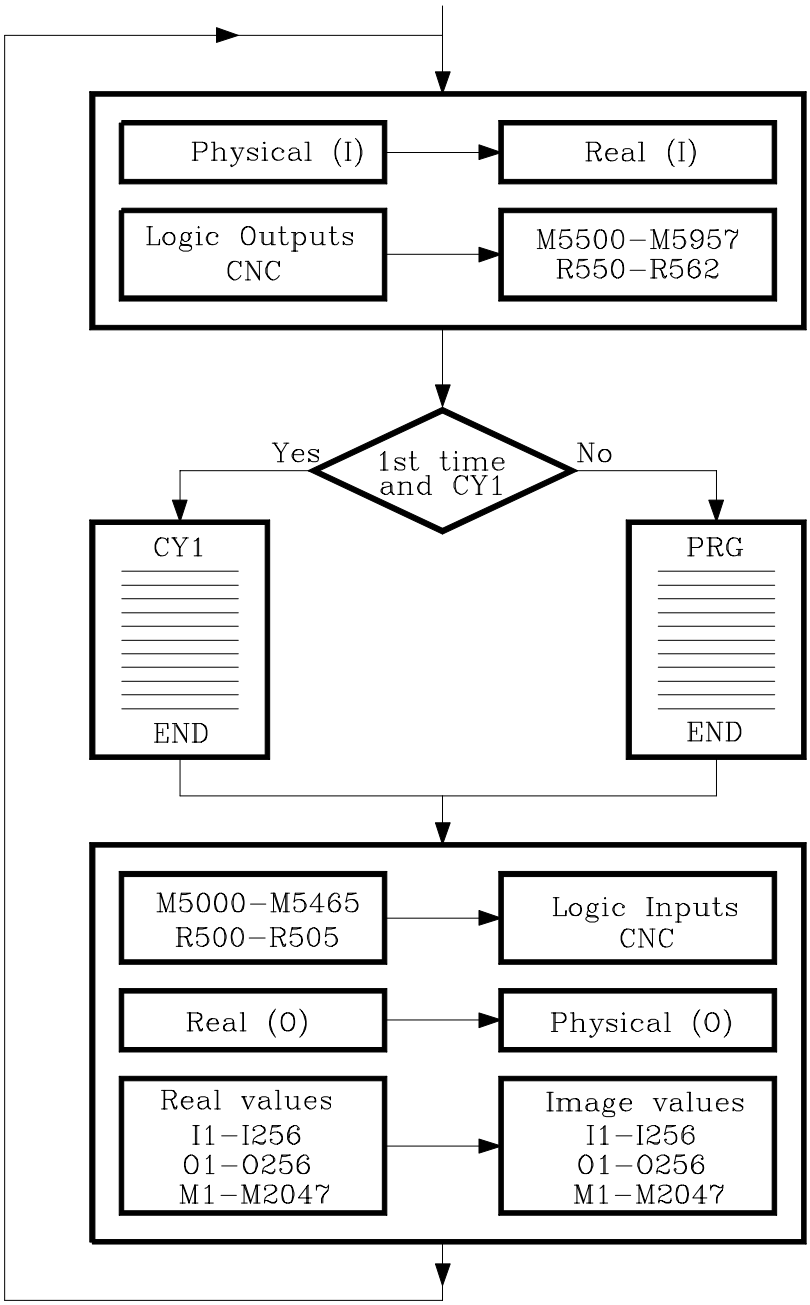
* It updates the values of resources M5000 thru M5957 and R500 thru R559 with the values of the CNC logic outputs (internal variables).

* Executes the program cycle.

* It updates the CNC logic inputs (internal variables) with the real values of resources M5000 thru M5957 and R500 thru R559.

* It assigns to the physical outputs (electrical cabinet) the real values of the corresponding "O" resources.

* It copies the real values of resources I, O, M into their own images.



If the PLC is stopped, it will work as follows:

* The real values of the "I" resources corresponding to the physical inputs will be updated every 10 milliseconds.

* The physical outputs will be updated every 10 milliseconds with the real values of the corresponding "O" resources.

* The PLC will attend to all requests and modifications of its internal variables.

9.4 ACTIVE MESSAGES

When selecting this option, the CNC will display a page (or screen) showing dynamically all the active messages generated by the PLC.

These messages will be listed by priority always starting from the one with the smallest number (highest priority).

The operator can move the cursor a line at a time with the up and down arrow keys or page by page with the page-up and page-down keys.

To delete one of the displayed messages, select it with the cursor and press the “DELETE MESSAGE” softkey.

Note that the CNC dynamically updates the active messages.

9.5 ACTIVE PAGES (SCREENS)

When selecting this option, the CNC will show the active page with the lowest number.

To delete a page or access the other active pages, the CNC will display the following softkey options:

NEXT PAGE	Press this softkey to display the next active page.
PREVIOUS PAGE	Press this softkey to display the previous active page.
CLEAR PAGE	Press this softkey to deactivate the page being displayed.

Note that the CNC dynamically updates the active pages.

9.6 SAVE PROGRAM

Press this softkey to save the PLC_PRG program into the user "Memkey Card" (CARD A).

The PLC program must be stopped before attempting to save it. If it is running, the CNC will ask whether it is desired to stop it or not.

The PLC program must be compiled, otherwise, the CNC will issue an warning message

If the PLC program is running, the CNC requests it to be stopped.

9.7 RESTORE PROGRAM

Press this softkey to restore (recover) the PLC program (PLC_PRG) from the user "Memkey Card" (CARD A) .

The PLC program must not be running any PLC program, otherwise, the CNC will ask whether it is desired to stop it or not.

After executing this instruction, the new source program recovered will replace the one that the PLC previously had. This new one must be compiled and started in order for the PLC to execute it.

9.8 RESOURCES IN USE

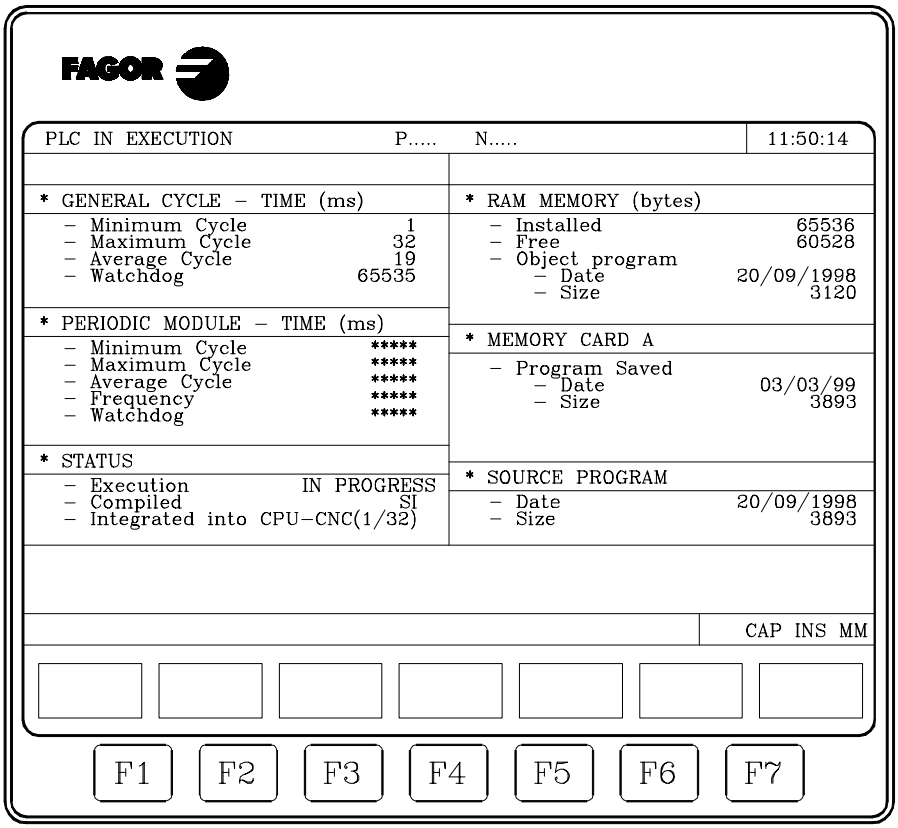
When selecting this option, the CNC will offer the softkeys to select the table of resources used in the PLC program.

The following resource tables are available:

INPUTS (I)
OUTPUTS (O)
MARKS (M)
REGISTERS (R)
TIMERS (T)
COUNTERS (C)

9.9 STATISTICS

This option shows the PLC memory distribution, the execution time of the various PLC modules, the PLC program status and the date when it was edited.



GENERAL CYCLE

This section shows the time (maximum, minimum and average) it takes the PLC to execute a program cycle.

This cycle includes:

* Updating the resources with the values of the physical inputs and internal CNC variables.

* Executing both the main cycle (PRG) and the periodic module.

* Updating the internal CNC variables and the physical outputs with the resource variables.

* Copying the resources into their corresponding images.

This section also shows the watchdog time selected by the PLC machine parameter “WDGPRG”.

PERIODIC MODULE

This section shows the time (maximum, minimum and average) that it takes to execute the periodic module of the PLC.

It also shows the period assigned to this module by means of the directive instruction “PE t”. This period indicates how frequently the periodic module will be executed (every “t” milliseconds).

It also shows the watchdog time for this module selected by the PLC machine parameter “WDGPER”.

STATUS

Provides information on the PLC program status indicating whether it is compiled or not and whether it is stopped or in execution.

When the PLC does not have its own CPU (integrated into CPU-CNC), it will also indicate the time that the CNC’s CPU dedicates to the PLC. This value Will defined by the PLC machine parameter “CPUTIME”.

RAM MEMORY

This section indicates the system’s RAM memory available for the exclusive use of the PLC (installed) and it also indicates how much free memory there is.

The object program (executable) is obtained when compiling the source program and is the one executed by the PLC. This section shows the date when it was generated and the RAM memory space it occupies (size).

MEMORY CARD A

This section also shows the date the PLC program (PLC_PRG) was saved into the "Memkey Card" and its size.

SOURCE PROGRAM

This section indicates the date when it was last edited and its size.

The PLC source program is stored in the CNC’s RAM memory.

9.10 LOGIC ANALYZER

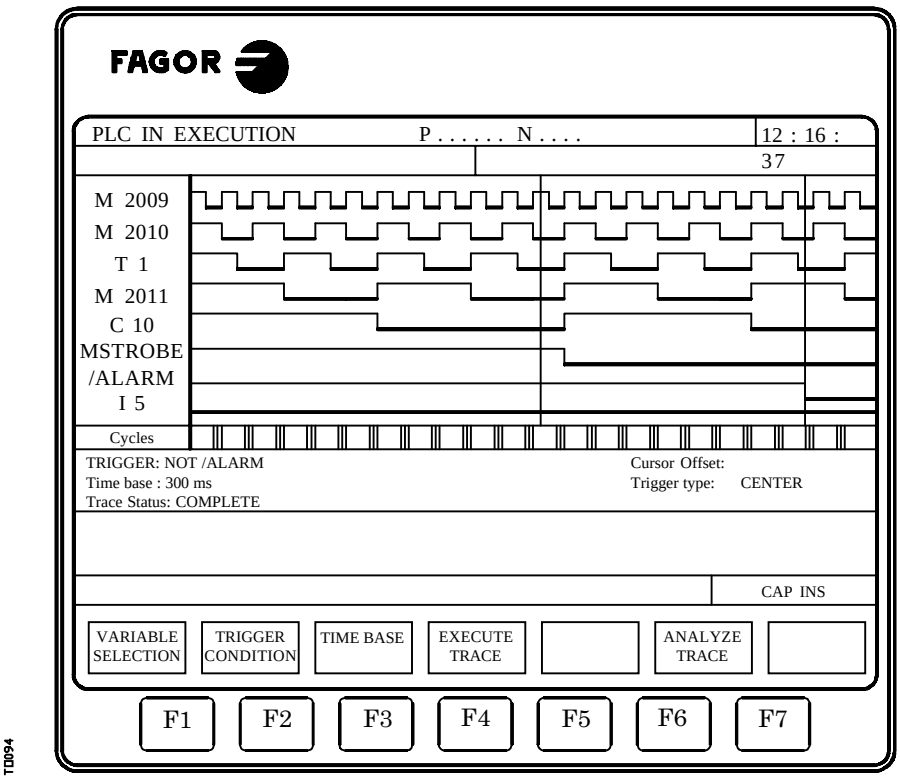
The logic analyzer is especially indicated to perform the machine setup and to determine errors and critical situations in the behavior of the various signals.

With this option it is possible to analyze the behavior of the logic signals of the PLC according to a time base and some trigger conditions established by the user.

Up to 8 signals can be monitored simultaneously. The results are displayed using a graphic interface to simplify the interpretation of the obtained data.

9.10.1 DESCRIPTION OF THE WORK SCREEN

The screen for the logic analyzer can be divided into the following display windows or areas:



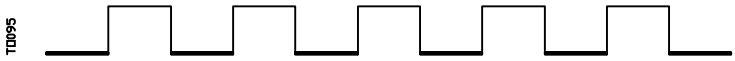
TD094

1.- Status window

It displays the graphic representation of the status of each one of the selected signals.

* The variable area shows the names or symbols of the logic signals to be analyzed.

* The status area shows the status of each variable in the shape of square waves. The line corresponding to logic level 0 is shown with a thicker line.



Also, a vertical red line is displayed to indicate the TRIGGER point and a vertical green line indicating the cursor position.

The green cursor line can be slid right and left along the trace and it can be used to measure the time difference between two of its points.

The status area is divided in several vertical sections. Each of them represents the amount of time established by the "time base" constant.

This constant determines the resolution of the logic signals and, after being defined by the user, can be modified at will. The relationship between the "time base" and the signal resolution is inversely proportional in such way that the smaller the time base, the greater the signal resolution is and vice versa.

2.- Cycle window

This window displays a series of vertical lines "|". Each one of them indicates the instant when a new PLC program cycle starts being executed.

It allows to maintain a relationship between the flow of the logic signals and the duration of each PLC execution cycle.

3.- Information window

This window provides general information about the trace being shown at the time. The shown data is the following:

Trigger — It shows the trigger condition set by the user to do the trace.

Time Base — Indicates the time base set by the user and used to show the current trace.

Trace Status — Indicates the current trace status. The shown texts and their meanings are as follows:

Empty	There is no calculated trace.
Capturing	There is one trace in progress.
Complete	One stored trace is available.

Cursor Offset Indicates the time difference, in milliseconds, between the cursor position (green line) and the trigger position (red line).

Trigger Type Indicates the type of trigger selected. The texts shown and their meanings are the following:

Before	The trigger is positioned at the beginning of the trace.
After	The trigger is positioned at the end of the trace.
Center	The trigger is positioned at the center of the trace.
Default	When no trigger condition has been specified.

4.- Editing window

It is the standard CNC editing window. It is used for all the processes requiring data entry.

5.- Message window

The CNC uses this window to display a warning or error message.

9.10.2 SELECTION OF VARIABLES AND TRIGGER CONDITIONS

Before requesting a trace, it is necessary to define the variables to be analyzed, the trigger type and conditions and the time base to be used to display the captured data.

To do this, the following softkey options are available: "**VARIABLE SELECTION**", "**TRIGGER CONDITION**" and "**TIME BASE**".

9.10.2.1 VARIABLE SELECTION

With this option it is possible to select up to 8 variables to be analyzed later.

It displays a cursor over the variable area and it can be slid up and down by means of the up and down arrow keys. The following softkey options will appear:

EDIT

With this option it is possible to edit a new variable or modify one of the currently defined variables.

Before pressing this softkey, we must select, with the cursor, the location where that variable will be shown.

Once this option is selected, the softkeys will change their background color to white and they will show the information corresponding to the editing type possible.

It is possible to analyze any logic signal of the PLC (I3, B1R120, TEN 3, CDW 4, DFU M200, etc.) and it can be referred to by its name or by its associated symbol.

It is also possible to analyze logic expressions, formed with one or more consultations which must follow the syntax and rules used to write the PLC equations.

M100 AND (NOT I15 OR I5) AND CPS C1 EQ 100

Although it might seem difficult to understand the processing of expressions and consultations at a logic analyzer, it should be borne in mind that it could prove very useful when it comes to finding out the status of a whole expression..

It is **not** possible to use more than 16 flank (edge) detecting instructions (DFU and DFD) among all the selected variable definitions and trigger conditions.

By pressing the **ESC** key, the variable being edited will be deleted. From this point on, that variable can be edited again.

Once the variable has been edited, press the **ENTER** key. The new variable will appear in the cursor position inside the variable area.

Only the first 8 characters of the selected variable or expression are shown even when it has more than 8.

The cursor will position at the next variable which will be shown in the editing window, thus being possible to continue editing new variables.

To quit this option, the editing area must be empty. If it is not empty, delete its contents by pressing **ESC** and then press **ESC** again.

DELETE

Use this option to delete a variable.

Before pressing this softkey, use the cursor to select the variable to be deleted.

To delete more variables, repeat these steps for each one of them.

CLEAR ALL

This option deletes all variables from the status window.

9.10.2.2 SELECTION OF TRIGGER CONDITION

The trigger condition as defined is that around which the data capture takes place. This data capture can be done before after or both before and after having met the selected trigger condition.

With this option it is possible to select the trigger type and condition of the logic analyzer. To do this, the following softkey options appear:

EDIT

With this option it is possible to edit the trigger condition around which the data capture will take place.

Once this option is selected, the softkeys will change their background color to white and they will show the information corresponding to the editing type possible.

It is possible to analyze logic expressions, formed with one or more consultations which must follow the syntax and rules used to write the PLC equations.

Examples of expressions and trigger conditions:

M100	The trigger occurs when M100 = 1
NOT M100	The trigger occurs when M100 = 0
CPS R100 EQ 1	The trigger occurs when R100 = 1
NOT I20 AND I5	The trigger occurs when the expression is true

It is **not** possible to use more than 16 flank (edge) detecting instructions (DFU and DFD) among all the selected variable definitions and trigger conditions.

By pressing the **ESC** key, the trigger condition being edited will be deleted. From this point on, that condition can be edited again.

Once the trigger condition has been edited, press ENTER. The new trigger condition will appear at the information window.

If no trigger condition has been specified, the system assumes one by default and it displays the message: "Trigger type: DEFAULT" in the information window. Besides, it will not permit the selection of any other possible types of trigger (before, center or after).

TRIGGER BEFORE

The CNC starts the data capture once (**after**) the selected trigger condition is met.

Then, once the trace has been executed, the trigger (vertical red line) will be positioned at the beginning of the trace.

TRIGGER AFTER

The CNC starts the data capture at the very instant the user selects the option to execute the trace **(before** the trigger condition is met).

The trace will be considered done when the selected trigger condition is met.

The trigger (vertical red line) will be positioned at the end of the trace.

TRIGGER CENTER

The CNC starts the data capture at the very instant the user selects the option to execute the trace.

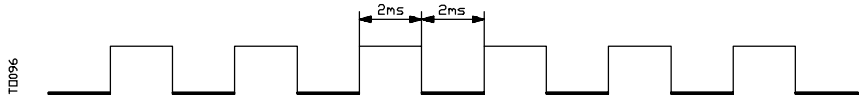
Then, once the trace has been executed, the trigger (vertical red line) will be positioned in the center of the trace.

9.10.2.3 SELECTION OF TIME BASE

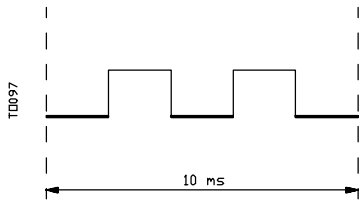
By means of this parameter, the user specifies the amount of time represented by each of vertical intervals.

Since the CRT width of these intervals is always the same, the signal resolution will be established by this time base in such way that the smaller the time base, the greater the signal resolution will be.

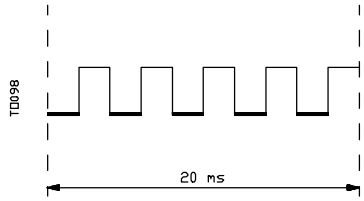
Example: Having a Mark whose status changes every 2 milliseconds.



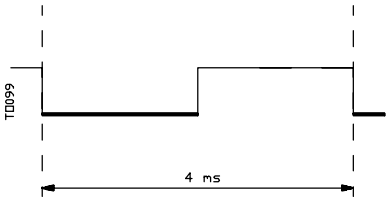
With a time base of 10 milliseconds, it will appear as follows:



With a time base of 20 milliseconds, it will appear as follows:



With a time base of 4 milliseconds, it will appear as follows:



The time base is given in milliseconds and the information window will show the selected value. By default, the CNC assumes a time base of 10 milliseconds.

It is possible to set a time base equal to the frequency of the signal to be monitored and then change it to obtain a finer signal resolution when analyzing the trace.

9.10.3 EXECUTE TRACE

Once having selected the variables and trigger conditions desired, press the "**EXECUTE TRACE**" softkey to indicate to the CNC to begin the data capture.

When the selected trigger condition is met, the trigger line displayed at the information window will change its color.

While the trace is being executed, the information window will display the message: "Trace Status: CAPTURING".

The trace will be completed when the internal memory buffer, dedicated to this function, is full or it is interrupted by pressing the "STOP TRACE" softkey. At this point, the information window will show the message: "Trace Status: COMPLETE".

9.10.3.1 DATA CAPTURE

The data capture takes place at the beginning of each cycle (PRG and PE), after reading the physical inputs and updating the marks corresponding to the CNC logic outputs and just before starting the PLC program execution.

Use this instruction to carry out another data capture while executing the PLC cycle.

This instruction permits the data capture of signals changing at frequencies greater than the cycle time as well as of those changing status during the execution of the cycle while keeping it the same at the beginning and at the end of the cycle.

Example of how to use the "**TRACE**" instruction:

```
PRG
   ----------
   ----------
   TRACE        ; Data capture
   ----------
   ----------
   TRACE        ; Data capture
   ----------
   ----------
   TRACE        ; Data capture
   ----------
   ----------
END

PE5
   ----------
   TRACE        ; Data capture
   ----------
END
```

The data capture in the execution of the trace in this program takes place:

- At the beginning of each **PRG** cycle
- Every time the periodic cycle (**PE**) is executed (every 5 milliseconds)
- 3 times while executing the **PRG** module.
- Once while executing the **PE** module.

This way, by means of the "**TRACE**" instruction the data capture can be done any time, especially at those program points considered more critical.

This instruction must only be used when debugging the PLC program and it should be avoided once the PLC program is fully debugged.

9.10.3.2 MODES OF OPERATION

The way the data is captured depends on the type of trigger selected. This section describes the different types of trigger being used as well as the way the data capture is done in each case.

Trigger Before

The data capture begins as soon as the selected trigger condition is met, that is when the trigger line shown at the information window changes its color.

The trace will be completed when the trace buffer is full or when the user interrupts it with the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, the trace will be empty.

Trigger after

The data capture begins the instant the user presses the "**EXECUTE TRACE**" softkey.

The trace will be completed when the selected trigger condition is met or it is interrupted by pressing the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, a trace will be shown with data but without the trigger position (vertical red line).

Trigger center

The data capture begins the instant the user presses the "**EXECUTE TRACE**" softkey.

The CNC will enable half the trace buffer to store the data corresponding to the trace prior to the trigger and the other half for the data corresponding to the trace after the trigger.

The trace is completed when its buffer is full or when it is interrupted by pressing the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, a trace will be shown with data but without the trigger position (vertical red line).

Trigger by Default

The CNC carries out this type of trace when no trigger condition has been specified.

The data capture begins the instant the "**EXECUTE TRACE**" softkey is pressed.

The trace is completed when interrupted by pressing the "**STOP TRACE**" showing a trace with data but without the trigger position (vertical red line).

9.10.3.3 TRACE REPRESENTATION

Once the data capture is done, the CNC will display graphically in the status window the status of the signals based on the trace calculated for the analyzed variables.

Also, a vertical red line indicating the trigger position and a vertical green line indicating the cursor position will appear superimposed on the trace.

The cursor position (vertical green line) can be slid along the trace by means of the following keys:

Left arrow Moves the cursor one pixel to the left.

While keeping this key pressed, the cursor will advance automatically one pixel at a time and increasing its speed.

If the cursor is positioned at the left end, the trace will be shifted to the right while the cursor stays in the same position.

Right arrow Moves the cursor one pixel to the right.

While keeping this key pressed, the cursor will advance automatically one pixel at a time and increasing its speed.

If the cursor is positioned at the right end, the trace will be shifted to the left while the cursor stays in the same position.

Previous page Moves the cursor one screen to the left.

Next page Moves the cursor one screen to the right.

The CNC will show at all times, in the information window, the cursor position (vertical green line) with respect to the trigger position (vertical red line). This information will appear as “Cursor Offset” and it will be given in milliseconds.

9.10.4 ANALYZE TRACE

Once the data capture is done, the CNC, besides displaying the status window, will enable the "**ANALYZE TRACE**" softkey.

With this option it is possible to position the cursor (vertical green line) at the beginning of the trace, at the end of it or at a specific point along the trace. It is also possible to change the time base for the trace or calculate the time difference between two points of the trace.

To do this, the following softkey options are available:

Find beginning The cursor will position at the beginning of the trace being shown.

Find End It will show the last section of the trace and the cursor will position at the end of it.

Find Trigger It will show the area of the trace corresponding to the trigger zone. The trigger position will appear as a vertical red line over the trace.

The CNC will execute this option when a trigger occurs while analyzing the trace.

Find Time Base When pressing this key, the CNC will request the cursor position with respect to the trigger point. This value is given in milliseconds.

For example: Having selected a "Find time base" of -1000 milliseconds, the CNC will show the trace section corresponding to 1 second prior to the trigger instant.

If no trigger occurred while analyzing the trace, the CNC will assume that the indicated position is referred to the beginning of the trace.

Calculate Times With this option it is possible to find out the time between two points of the trace. To do this, follow these steps in order to set the initial and final points of the calculation.

Position the cursor at the initial point of calculation and press the "**MARK BEGINNING**" softkey to validate it. Use the "left arrow", "right arrow", "page-up" and "page down" keys to move the cursor.

Position the cursor at the final point of calculation and press the "**MARK END**" softkey to validate it.

The CNC will display in the message window the time difference between those two points. It will be given in milliseconds.

This feature can prove very useful to calculate exactly the rise and fall times of a signal, times between two signals, times between the trigger of a signal and the beginning of a cycle, etc.

Modify Time Base This option permits the "Time Base" to be modified.

The status area is divided into several vertical sections. Each of these sections represents a time pitch determined by the "Time Base" constant.

The relationship between the "Time Base" and the signal resolution is inversely proportional in such way that the smaller the "time base", the greater the signal resolution and vice versa.

When pressing this softkey, the CNC will request the new value for the time base. This value must be given in milliseconds.

10. SCREEN EDITOR

In this operating mode, the operator can create up to 256 pages (screens) which will be stored in the "Memkey Card".

The operator can also create up to 256 SYMBOLS to be used when creating the user screens. These symbols are also stored in the "Memkey Card".

The information contained in a page or symbol cannot occupy more than 4Kb of memory. Otherwise, the CNC will issue the corresponding error message.

The user screens stored in the "Memkey Card" may be:

* Used in the screen customizing programs as described next.

* Displayed on power-up (page 0) instead of the FAGOR logo.

* Activated from the PLC.

  The PLC has 256 marks, with their corresponding mnemonics, to select the user screens. These marks are:

M4700	PIC0
M4701	PIC1
M4702	PIC2
———	———
———	———-
M4953	PIC253
M4954	PIC254
M4955	PIC255

  When any of these marks is set high, its corresponding screen (page) is activated.

* Used to complete the M function assistance system (screens 250-255).

  When requesting programming assistance for the auxiliary M functions by pressing the [HELP] key, the CNC will show the corresponding internal screen (page).

  When user page 250 is defined, that information will also include the symbol indicating that more help pages are available. By pressing this key, the CNC will display user screen 250.

  The CNC will keep showing that indicator as long as there are more user screens defined (250-255).

  These screens must be defined in a row always starting from page 250. If one of them is missing, the CNC will interpret that there are no more screens defined.

The user screens activated from the PLC may be displayed with the ACTIVE PAGES option of the PLC.

The various options available in this operating mode are:

* **UTILITIES** to manipulate user symbols and screens (edit, copy, delete, etc.).

* **GRAPHIC ELEMENTS** to insert graphic elements in the selected symbol or screen.

* **TEXTS** to insert texts in the selected symbol or screen.

* **MODIFICATIONS** to modify the selected symbol or screen.

10.1 UTILITIES

The various options available in this mode are:

DIRECTORY

To display the directory of user screens and symbols that are stored in the "Memkey Card" (CARD A) or in external devices through the serial lines.

Select the desired device and directory.

The CNC shows the size (in bytes) of each user screen (page) and symbol.

COPY

To make copies within the "Memkey Card" (CARD A) or between the "CARD A" and the external devices.

Examples:

to copy screen (page) 5 from the "Memkey Card" to serial line 2

COPY PAGE 5 IN SERIAL LINE 2 (DNC)

to copy screen (page) 50 from serial line 2 into the "Memkey Card"

COPY SERIAL LINE 2 (DNC) IN PAGE 50 ENTER

to copy symbol 15 as symbol 16 within the "Memkey Card"

COPY SYMBOL 15 IN SYMBOL 16 ENTER

DELETE

To delete a screen or symbol from the "Memkey Card". To do that, proceed as follows:

- Press the [DELETE] softkey
- Press the [PAGE] or [SYMBOL] softkey
- Key in the page or screen number to be deleted and press [ENTER]

The CNC will request confirmation of the command.

RENAME

To assign a new name or comment to a page or symbol of the "Memkey Card".

If there is another one with the same number, the CNC will display a warning message and it will offer the chance to modify the command.

Examples:

to change the page number from 20 to 55

RENAME PAGE 20 TO NEW NUMBER 55 ENTER

to change the comment of symbol 10

RENAME SYMBOL 10 TO NEW COMMENT "Test" ENTER

EDIT

To edit a new user screen (page) or symbol proceed as follows:

- Press the [EDIT] softkey
- Press the [PAGE] or [SYMBOL] softkey
- Key in the page or symbol number
- Press [ENTER]

If the page or symbol does not exists, an empty page will appear in the editing area.

How to edit user screens and symbols is described later on in this chapter.

If the selected screen or symbol has been changed, the CNC will request whether it is to be saved or not in the following instances:

- When exiting the screen editor.
- When selecting another screen (page) or symbol.

SAVE

To save the page or symbol being edited into the "Memkey Card".

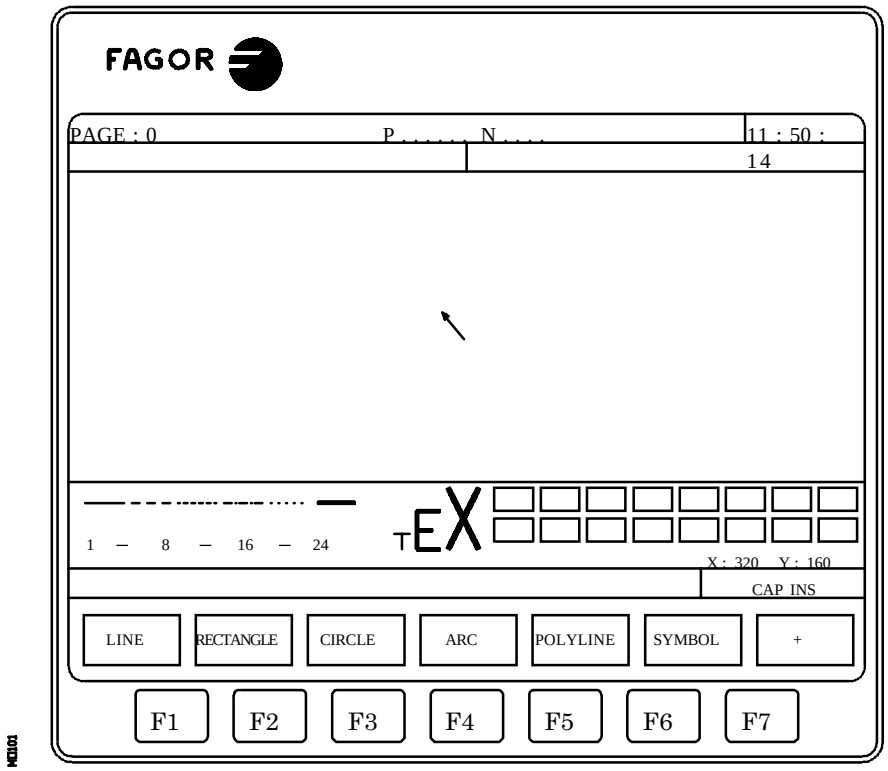
10.2 EDITING CUSTOM SCREENS (PAGES) AND SYMBOLS

In order to edit a page or symbol, it is necessary to selected first by means of the EDIT option of the UTILITIES mode of operation.

To edit or modify a page or symbol, use the options: **GRAPHIC ELEMENTS, TEXTS**, and **MODIFICATIONS**.

The information contained in a page or symbol must not occupy more than 4Kb; otherwise, the CNC will issue the corresponding error message.

Once the page or symbol has been selected, the CNC will display a screen similar to this



one:

* The upper left-hand side of the screen will show the number of the page or symbol being edited.

* The main window will show the selected page or symbol. When it is a new page or symbol, the main window will be “blank” (blue background).

* There is also a window at the bottom of the screen which shows the different editing parameters and highlights their selected values.

The various parameters available are:

* The type of drawing line used when defining the graphic elements.

* The cursor moving steps (cursor advance) in pixels.

* The letter size to create the texts for the pages and symbols.

* The background and foreground (main) colors for the graphic elements and for the letters.

  One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

This window also shows The cursor position coordinates in pixels. The horizontal position is indicated by the X value (1 through 638) and the vertical position by the Y value (0 through 334).

Once one of the options (**GRAPHIC ELEMENTS, TEXTS** or **MODIFICATIONS**) has been selected, it will be possible to modify the editing parameters any time.

This way, it will be possible to edit texts and shapes of different color and size.

Press **INS** to access this menu.

Once in this mode, the CNC will show the softkeys corresponding to the various options to modify these parameters. These options are described next.

Press **INS** again to quit this mode and return to the previous menu.

CURSOR ADVANCE

With this option it is possible to select the cursor moving step in pixels (1, 8, 16, 24).

Follow these steps after pressing this softkey:

1.- Use the right and left arrow keys to select the desired step.

The currently selected step will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the default value of 8.

TYPE OF LINE

With this option it is possible to select the type of line used to define the graphic elements.

Follow these steps after pressing this softkey:

1.- Use the right and left arrow keys to select the desired type of line.

The currently selected line type will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the “fine line” by default.

It is not possible to use the thick line to draw polylines or polygons. They are always drawn in fine line.

TEXT SIZE

With this option it is possible to select the size of the letters used to write the texts to be inserted in the pages or symbols.

Three sizes are available:

* Normal size.

  All the characters of the keyboard, numbers, signs, upper and lower case letters, can be written in this size.

* Double and triple sizes.

  Only capital letters **A through Z**, numbers **0 through 9** ; the “*****”, “**+**”, “**-**”, “**.**”, “**:**” , "**#**", "**%**", "**/**", "**<**", "**>**", "**?**" signs and the special characters: "**Ç**", "**Ä**", "**Ö**", "**Ü**", "**ß**" can be written in these sizes.

  When selecting lower case letters for these sizes, the CNC will convert them automatically into upper case.

Follow these steps to select the text size after pressing this softkey:

1.- Use the right and left arrow keys to select the desired size.

The currently selected size will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the normal size by default.

BACKGROUND COLOR

With this option it is possible to select the background color over which the different graphic elements and texts will be edited.

It is not possible to select the background color when editing a symbol since it is an attribute of the page and not of the symbol. Therefore, when inserting a symbol into a page, the symbol will take the background of that page.

If the desired background color is **WHITE**, it is recommended to use a different color while creating the page since the cursor the "drawing" cursor is always white and will become invisible with this background color. Once the complete page (screen) is created, the background color can be changed to the desired one.

One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

To select the background color, follow these steps:

1.- Use the right and left arrow keys to select the desired color among the 16 shown.

The CNC will show the background color being selected by placing the main-color rectangle inside the rectangle corresponding to the background color being selected.

2.- Press **ENTER** to validate the selected color or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes a blue background color by default.

MAIN COLOR

With this option it is possible to select the color used to draw and write texts on the page (screen) or symbol.

One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

To select the main color, follow these steps:

1.- Use the right and left arrow keys to select the desired color among the 16 shown.

The CNC will show the main color being selected by placing a white inside rectangle. It will also display the rectangle containing both the selected background color and the main color being selected here.

2.- Press **ENTER** to validate the selected color or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes white as the main color by default.

GRID

This softkey superimposes a grid over the screen in order to facilitate the lay out of the different components of the page or symbol being created or modified. This grid is formed by white or black points (depending on the background color) separated 16 pixels from each-other.

The grid points will be white when the selected background color corresponds to one of the 8 upper color rectangles and they will be black when the selected background color corresponds to one of the 8 lower color rectangles.

Press this softkey again to get rid of the grid.

Every time the grid is displayed, the CNC will reset the cursor advance (step) to 16 pixels.

Therefore, the cursor will move from grid point to grid point every time the arrow keys are pressed to position it on the screen. However, the cursor advance may be modified afterwards by selecting it with the **CURSOR ADVANCE** softkey.

10.3 GRAPHIC ELEMENTS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include graphic elements in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT → Positions the cursor at the last column (X638)

SHIFT ← Positions the cursor at the first column (X1)

SHIFT ↑ Positions the cursor at the first row (Y0).

SHIFT ↓ Positions the cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the CNC will position the cursor at the indicated coordinates.

Once this option is selected, it is possible to modify the editing parameters at any time even while defining the graphic elements. This way, it is possible to edit shapes of different line and color.

Press **INS** to access this menu.

Once in this mode, press the corresponding softkey to modify those parameters.

Press **INS** again to quit this mode and return to the previous menu.

The possible graphic elements which can be used to create a page or symbol are selected with the softkeys and are the following:

LINE

Follow these steps after pressing this softkey:

1.- Place the cursor at the beginning of the line and press **ENTER** to validate it.

2.- Move the cursor to the end of the line (the CNC will continuously show the line being drawn).

3.- Press **ENTER** to validate the line or **ESC** to cancel it.

Repeat the preceding steps to draw more lines. If no more lines are desired, press **ESC** to return to the previous menu.

RECTANGLE

Follow these steps after pressing this softkey:

1.- Place the cursor on one of the corners of the rectangle and press **ENTER** to validate it.

2.- Move the cursor to the opposite corner. The CNC will continuously show the rectangle being drawn.

3.- Press **ENTER** to validate the rectangle or **ESC** to cancel it.

Repeat these steps to draw more rectangles. If no more rectangles are desired, press ESC to return to the previous menu.

CIRCLE

Follow these steps after pressing this softkey:

1.- Place the cursor at the center of the circle and press **ENTER** to validate it.

2.- Move the cursor in order to define the radius. As the cursor moves, the CNC will show the circle corresponding to that radius.

3.- Press **ENTER** to validate the circle or **ESC** to cancel it.

Once the circle is validated, the cursor is positioned at its center in order to facilitate the drawing of concentric circles.

Repeat these steps to draw more circles. If no more circles are desired, press **ESC** to return to the previous menu.

ARC

Follow these steps after pressing this softkey:

1.- Place the cursor at one of the arc's ends and press **ENTER** to validate it.

2.- Move the cursor to the other end of the arc (the CNC will show a line joining both ends) and press **ENTER** to validate it.

The cursor is now positioned automatically at the center of that line.

3.- Move the cursor to define the curvature. The line will become an arc passing through 3 points (the two ends and the cursor point).

4.- Press **ENTER** to validate it or **ESC** to cancel it.

Repeat these steps to draw more arcs. If no more arcs are desired, press ESC to return to the previous menu.

POLYLINE

A polyline consists of several lines where the last point of one of them is the beginning point for the next one.

Follow these steps after pressing this softkey:

1.- Place the cursor at one of the ends of the polyline and press **ENTER** to validate it.

2.- Move the cursor to the end of the first line (which will be the beginning of the next one). The CNC will continuously show the line being drawn.

Press **ENTER** to validate the line or **ESC** to quit this option (which will delete the complete polyline).

3.- Repeat steps 1 and 2 for the rest of the lines.

Note that the maximum number of lines in a polyline is 127.

Once the polyline is drawn, press **ENTER** again to validate it or **ESC** to quit this option deleting the complete polyline.

Repeat these steps to draw more polylines and if no more polylines are desired, press ESC to return to the previous menu.

SYMBOL

This option allows a symbol to be drawn in the page or symbol being edited.

After pressing this softkey, the following steps will be taken.

1.- Enter the number of the symbol to include in the page or symbol being edited and press the ENTER key to validate it.

The CNC will show the cursor situated at the reference point corresponding to the symbol (upper left hand corner of the symbol).

2.- Move the cursor to the position where it is required to place the symbol. In this move, only the cursor will move and not the symbol.

3.- Press the ENTER key to validate it or the ESC key if you wish to quit.

Once the symbol has been validated the CNC will show it in the place indicated.

4.- To include more symbols, repeat the above operations.

5.- Press the ESC key to quit and go back to the previous menu.

If a symbol is being edited this symbol cannot be included in itself. Therefore, if symbol 4 is being edited, any symbol can be included except symbol 4.

Warning:

If a symbol is deleted, the CNC will update all the pages or symbols that contain it because all the calls to it will remain active.

When displaying a page or symbol which has a call to a nonexistent symbol (deleted or not defined), that area of the page will appear blank.

If this symbol is edited again later, the new representation assigned to the symbol will appear in all the pages and symbols which contain a call to it.

POLYGON

A polygon is a closed polyline whose beginning and end points coincide.

After pressing the softkey, the following steps will be taken:

1.- Place the cursor on one of the vertices of the polygon and press the ENTER key to validate it.

2.- Move the cursor to the following vertex of the polygon (the CNC will show the line you are trying to draw).

Press the ENTER key to validate the line or the ESC key if you wish to abandon.

3.- Repeat step 2 for the remaining vertices.

Once all vertices are defined, press the ENTER key and the CNC will complete the polygon or the ESC key if you wish to quit.

The maximum number of sides on the polygon is limited to 127.

FILLED POLYGON

After pressing this softkey, follow the steps as in the POLYGON option, but in this case, after completing the definition of the polygon it will be filled with the color used for its definition.

FILLED CIRCLE

After pressing this softkey follow the steps as in the CIRCLE option, but in this case, after completing the definition of the circle it will be filled with the color used for its definition.

FILLED RECTANGLE

After pressing this softkey follow the steps as in the RECTANGLE option, but in this case, after completing the definition of the rectangle it will be filled with the color used for its definition.

10.4 TEXTS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include texts in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT [→] Positions the cursor at the last column (X638)

SHIFT [←] Positions the cursor at the first column (X1)

SHIFT [↑] Positions the cursor at the first row (Y0).

SHIFT [↓] Positions The cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the CNC will position the cursor at the indicated coordinates.

Once this option is selected, it is possible to modify the editing parameters at any time even while defining the graphic elements. This way, it is possible to edit texts of different size and color.

Press **INS** to access this menu.

Once in this mode, press the corresponding softkey to modify those parameters.

Press **INS** again to quit this mode and return to the previous menu.

It is also possible to insert one of the texts available at the CNC or a text previously keyed in by the user. To do this, the following softkey options are available:

USER DEFINED TEXT

Follow these steps to insert the desired text:

1.- Press **ENTER**.

The CNC will display a text editing window. The cursor within this window can be moved with right and left arrow keys.

2.- "Type" the desired text.

A rectangle will be displayed which will enlarge as the text is "typed" in the editing window thus indicating the screen space that this text will occupy.

Press **ESC** to cancel this option and the previous menu will be displayed.

3.- Press **ENTER** once the text has been correctly "typed in".

The typed text will remain in the editing window and the cursor will be positioned in the main window.

4.- Position the rectangle by moving the cursor.

5.- Press **ENTER** to validate this command and the text will replace the rectangle on the screen.

Note that once the text has been "entered", neither its size nor its color can be modified. Therefore, these options must be selected before pressing **ENTER**.

TEXT NUMBER

With this option it is possible to select a text used by the CNC itself in its various operating modes and insert it into the current page or symbol.

To insert one of these predetermined texts, follow these steps:

1.- Press the corresponding softkey.

The CNC will show a screen area to indicate the text number. The cursor may be moved within this area with the right and left arrow keys.

2.- Indicate the desired number by keying it in from the keyboard and press **ENTER**.

The CNC will display the text corresponding to this number and the rectangle indicating the screen space it occupies.

If another text is desired, key in the other number and press **ENTER** again.

Press **ESC** to quit this option without inserting the text and the CNC will show the previous menu.

3.- Once the desired text has been selected, press **ENTER**.

The typed text will remain in the editing window and the cursor will be positioned in the main window.

4.- Position the rectangle by moving the cursor.

5.- Press **ENTER** to validate this command and the text will replace the rectangle on the screen.

Observe that once the text has been "entered", neither its size nor its color can be modified. Therefore, these options must be selected before pressing **ENTER**.

Warning:

This application may be useful when the pages or symbols being edited are to be shown in other languages since the CNC will translate them into the chosen language.

Usually, when the texts are to be shown in one single language, it is more practical to simply write them up instead of searching them in a list of more than 1500 predetermined messages.

However, should anyone desire the printout of these predetermined texts, feel free to request it from Fagor Automation.

10.5 MODIFICATIONS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include texts in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT Positions the cursor at the last column (X638)

SHIFT Positions the cursor at the first column (X1)

SHIFT Positions the cursor at the first row (Y0).

SHIFT Positions the cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the cursor will be positioned at the indicated coordinates.

The possible options to modify a page or symbol are:

CLEAR PAGE

Allows the selected page or symbol to be deleted.

Once this softkey has been pressed, the CNC will request an OK before executing the indicated operation.

If this option is executed, the CNC will delete the page or symbol being edited, but it will keep in the "Memkey Card" the contents of that page or symbol the last time the "SAVE” command was executed.

DELETE ELEMENTS

This option allows an element of the displayed page or symbol to be selected and then deleted.

To do this follow these steps:

1.- Place the cursor in the position to delete an element and press the **ENTER** key to validate it.

An area of between ± 8 pixels from the position indicated will be analyzed.

If the element to be deleted is a filled circle or a filled polygon, the cursor must be positioned on a point on the circumference or external polygon (periphery).

2.- If any graphic element or text exists in this area, this will be highlighted and you will be asked if you wish to delete it.

Press the ENTER key to delete this element, otherwise the ESC key.

Should there be several elements in this area, the CNC will highlight them in succession and it will ask for confirmation before deleting any of them.

MOVE SCREEN

With this option it is possible to reposition the whole page (not its individual elements separately) and it can only be used to move pages and not symbols.

It allows the entire page to be moved with the right, left, up and down arrow keys.

The center of the page is taken as a reference for this movement.

To do this follow these steps:

1.- The CNC will show the page with the cursor placed in the middle of the screen.

2.- Move the cursor to the position to place the page reference point.

Press ESC to quit this option without making any changes and the CNC will show the previous menu.

Repeat these steps to perform more moves, otherwise, press ESC and the CNC will show the previous menu.

11. *MACHINE PARAMETERS*

In order for the machine tool to execute the programmed instructions correctly, the CNC must know specific data on the machine such as feedrates, accelerations, feedbacks, automatic tool changes, etc.

This data is determined by the manufacturer of the machine and must be stored in the machine parameter tables.

These tables may be edited in this work mode or copied into the "Memkey Card" or a PC as described later on.

The CNC has the following groups of machine parameters:

* General machine parameters
* Axis parameters (one table per axis)
* Spindle parameters
* RS-422 and RS-232-C serial port configurations
* Ethernet configuration parameters
* PLC parameters
* M miscellaneous functions
* Leadscrew error compensation (one table per axis)
* Cross Compensations between two axes (for example: Beam sag).

First, the general machine parameters must be set as by means of these the machine axes are defined and therefore the Axis Parameter tables.

It must also be defined whether the machine has cross compensation and between which axes, and the CNC will generate the corresponding cross compensation parameters.

By means of the general machine parameters, the table lengths for the Tool Magazine, Tools, Tool Offsets and the miscellaneous M functions are defined.

By means of the Axis Parameters it is defined whether the axis has Leadscrew error Compensation or not and the length of the corresponding table.

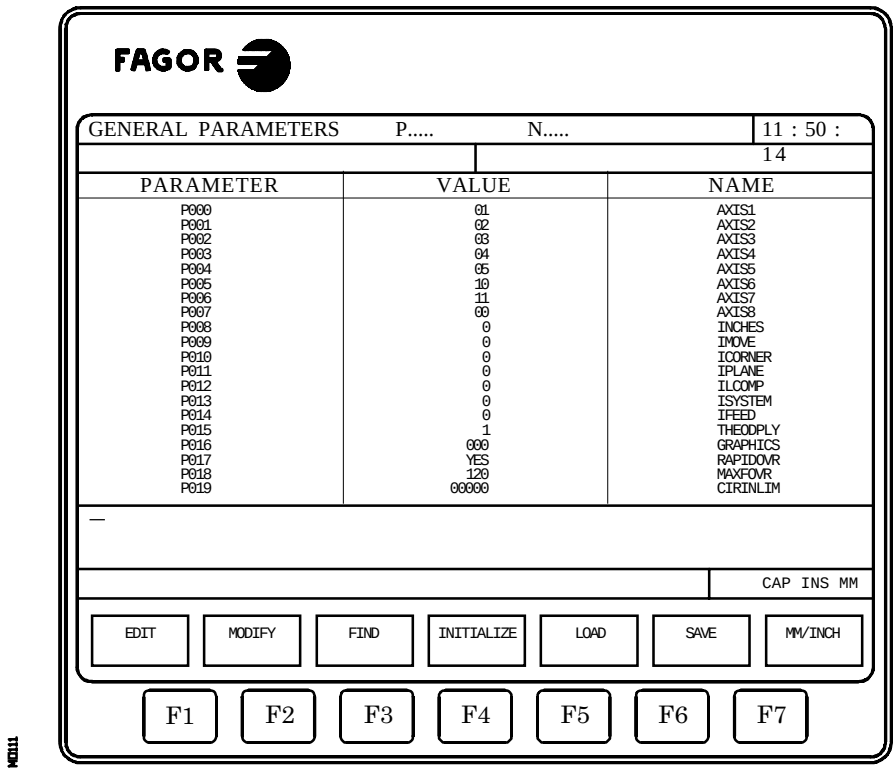
Once the general machine parameters are defined, press **SHIFT RESET** for the CNC to enable the required tables.

It is recommended to save the tables in the "Memkey Card" or out to a peripheral device or PC.

When accessing this operating mode, the CNC will show the tables that are saved in the "Memkey Card" (CARD A).

11.1 MACHINE PARAMETER TABLES

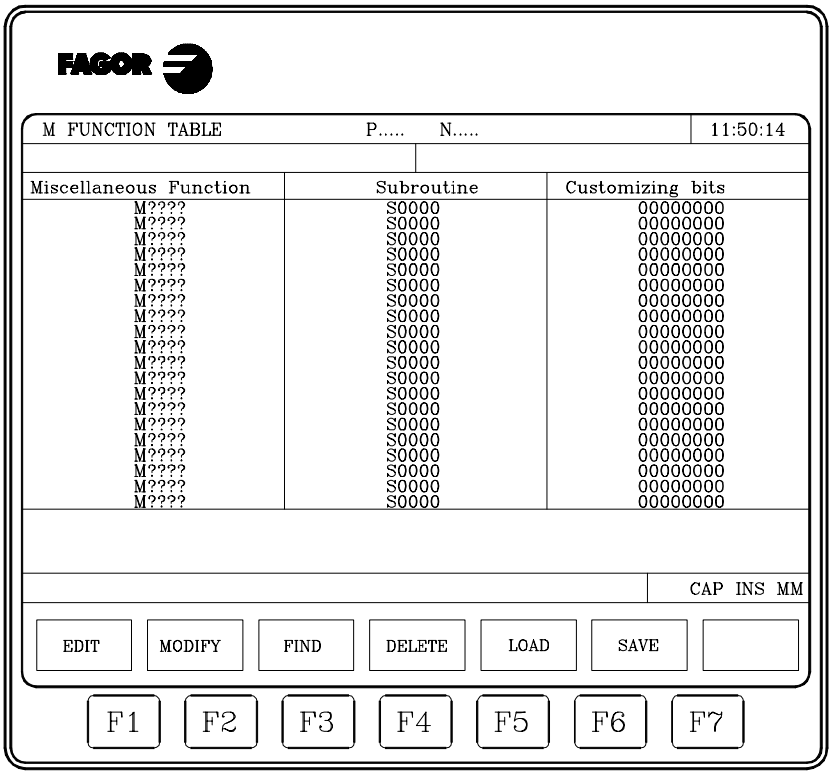
The General, Axis, Spindle, Serial ports and PLC tables have the following structure:



Where the parameter number is indicated, the value assigned to it and the name or mnemonic associated with this parameter.

11.2 MISCELLANEOUS FUNCTION TABLES

The table corresponding to the miscellaneous M functions has the following structure:



The number of M functions in the table is defined by means of the general machine parameter “NMISCFUN”. The following is defined for each line:

* The number (0-9999) of the defined miscellaneous M functions:

  If an M function is not defined, the CNC will show M????.

* The number of the subroutine to be associated with this miscellaneous function.

* 8 customizing bits

x	x	x	x	x	x	x	x
7	6	5	4	3	2	1	0

  Bit 0 Indicates whether the CNC must (=0) or must not (=1) wait for the signal AUXEND (signal of the M executed) to resume program execution.

  Bit 1 Indicates whether the M function is executed before (=0) or after (=1) the movement of the block in which it is programmed.

  Bit 2 Indicates whether the execution of the M function interrupts (=1) or not (=0) the preparation of the blocks.

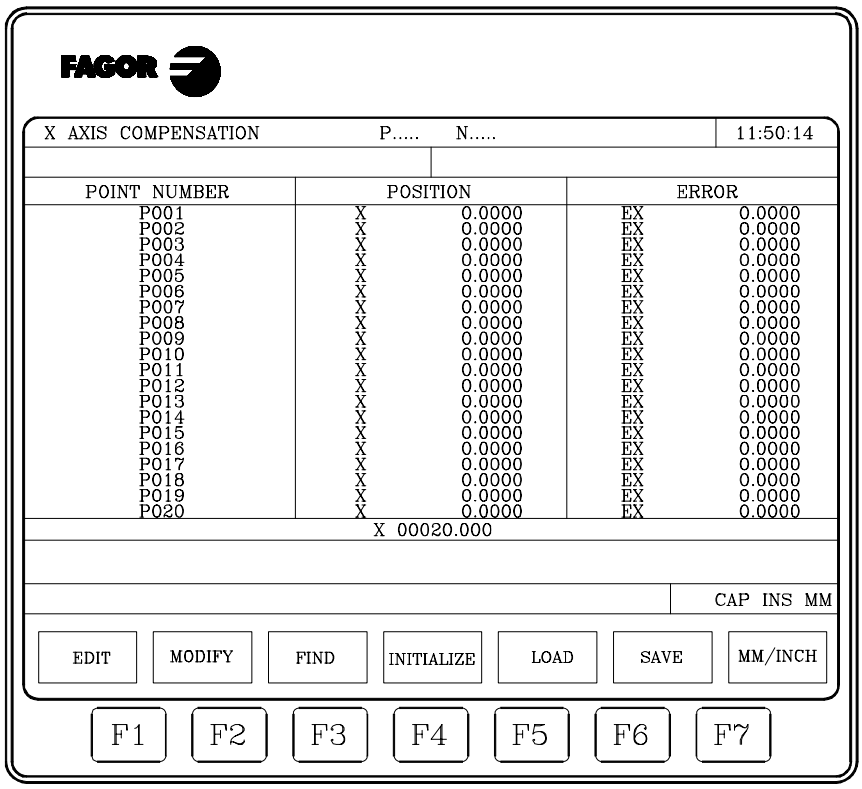
  Bit 3 Indicates whether the M function is executed after calling the associated subroutine (=0) or only the associated subroutine is executed (=1).

  Bit 4 When bit 2 is set to "1", it indicates whether block preparation is to be interrupted until the "M" function starts executing (=0) or until its execution is finished (=1).

  The rest of the bits are not being used at this time.

11.3 LEADSCREW ERROR COMPENSATION TABLES

The tables for leadscrew error compensation have the following structure:



The number of points of each of these is defined by means of the axis machine parameter "NPOINTS". The following is defined for each of line:

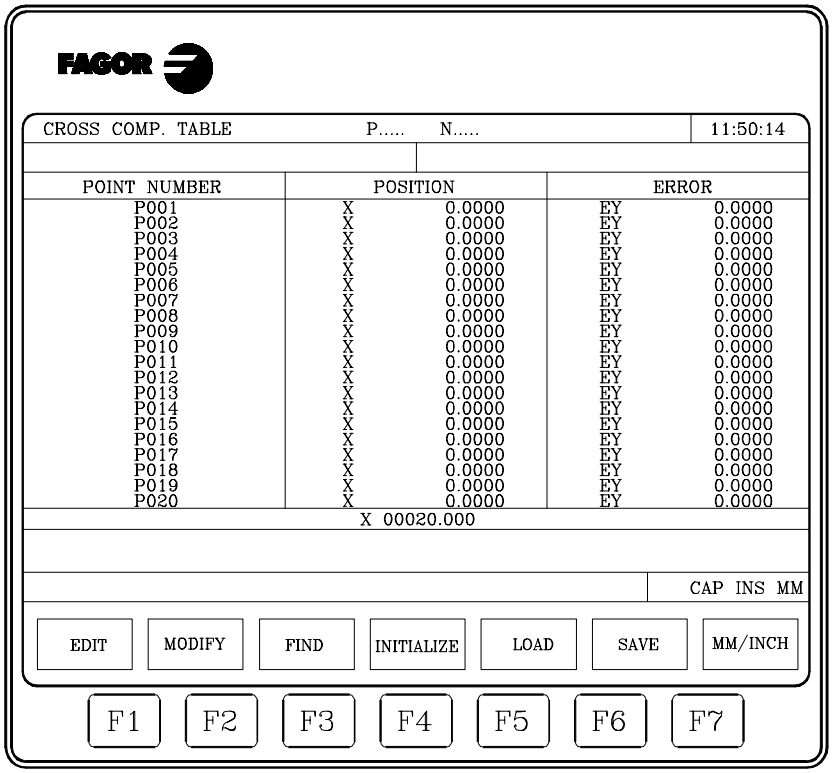
* Position of the axis to be compensated.

* Error of this axis in this position.

Also, the current position of the selected axis is displayed and updated as the machine axis moves.

11.4 CROSS COMPENSATION TABLES

The tables corresponding to cross compensation have the following structure.



The number of points of each table is defined by means of the general machine parameter “NPCROSS”, "NPCROSS2" and "NPCROSS3" respectively.

Each table defines:

* The position of the axis causing the error.

* The error suffered by the axis at that point.

Also, the current position of the selected axis is displayed. This position is updated as the axis moves.

11.5 OPERATION WITH PARAMETER TABLES

Once one of the tables has been selected, the cursor can be moved over the screen line by line by means of the “up and down arrow keys” or move from page to page by means of the “page up and page down keys”.

In addition, the user has an area of the screen for editing, it being possible to move the cursor over the screen by means of the “right arrow key and left arrow key”.

The CNC offers the following softkey options for each table:

EDIT

The desired parameter.

When selecting this option, the softkeys will change their color to a white background and they will show the various editing options.

In those tables corresponding to leadscrew and cross compensation, the position values of the axis must be edited as follows:

* Move the axis and when the error is found large enough to be considered, press the softkey corresponding to this axis.

* The CNC will include, in the editing area, the name of the axis followed by the position value corresponding to that point.

  This value can be modified if so desired.

* Press the softkey corresponding to the error and key in its value.

Once the parameter is edited, press **ENTER**. This new parameter will be included in the table and the cursor will be positioned over it. The editing area will be cleared, thus allowing other parameters to be edited.

Press **ESC** to quit this mode.

MODIFY

With this option it is possible to modify the selected parameter.

Before pressing this softkey, the desired parameter must be selected.

When selecting this option, the softkeys will change their color to a white background and they will show the various editing options.

By pressing **ESC**, the information displayed in the editing window (corresponding to the selected parameter) will be cleared. From this point on, a new value can be entered.

To quit this option, first clear the editing window using the **CL** key or the **ESC** key and then press **ESC** again. The selected parameter will not be modified.

Once this modification has concluded, press the **ENTER** key to validate it.

FIND

The beginning or end of the table, or the parameter whose number is indicated by positioning the cursor on the required parameter.

BEGINNING When pressing this softkey, the cursor positions over the first parameter of the table quitting this option.

END When pressing this softkey, the cursor positions over the last parameter of the table quitting this option.

PARAMETER When pressing this softkey, the CNC will request the number of the parameter to be found. Key in that number and press **ENTER**. The cursor will be positioned over the indicated parameter quitting this option.

INITIALIZE

With this option it is possible to reset all the parameters of the selected table to their default values.

These default values are indicated in the chapter corresponding to machine parameters in the installation manual.

LOAD

To load tables stored in the "Memkey Card" (CARD A) or in a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission begins after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the [ABORT] softkey.

If the length of the table received does not coincide with the length of the current table, the CNC will acts as follows:

If the table received is shorter than the current one, the received lines are modified and the rest keep their previous values.

If the table received is longer than the current one, the CNC updates all the lines of the current table and when detecting that there is no more room, the CNC issues the corresponding error message.

SAVE

The tables may be saved in the "Memkey Card" (CARD A) or in a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission begins after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the [ABORT] softkey.

MM/INCHES

Every time this softkey is pressed, the CNC will change the display format of those parameters affected by these units from millimeters to inches and vice versa.

The lower right-hand window will show the units currently selected.

Note that this change does not affect the general machine parameter "**INCHES**" which indicates the measuring units by default.

12. *DIAGNOSIS*

In this operating mode it is possible to know the configuration of the CNC as well as testing the system.

The CNC offers the following softkey options:

System Configuration
Hardware test
Memory test
Flash memory test
User
Hard disk

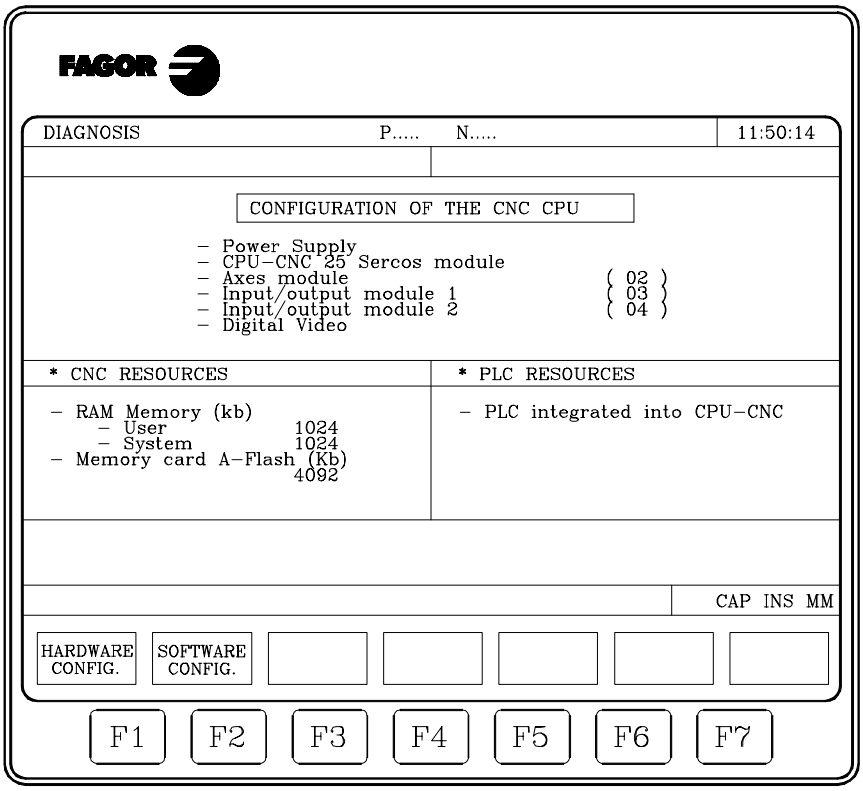
12.1 CONFIGURATION

This option shows the current system configuration.

Once this option has been chosen, two new softkeys will appear in order to select the hardware configuration or the software configuration of the system.

12.1.1 HARDWARE CONFIGURATION

This option shows the system configuration displaying the following information:



CONFIGURATION OF THE CENTRAL UNIT

It indicates the modules making up the new configuration of the central unit of the CNC.

The numbers which appear in brackets next to some of the modules and options indicate the logic address assigned to each of them.

CNC RESOURCES

It indicates the **RAM** memory (in Kb) available for the system and for the user.

It also indicates the memory of the "Memkey Card" in Kb.

12.1.2 SOFTWARE CONFIGURATION

This option shows:

* All available software options.

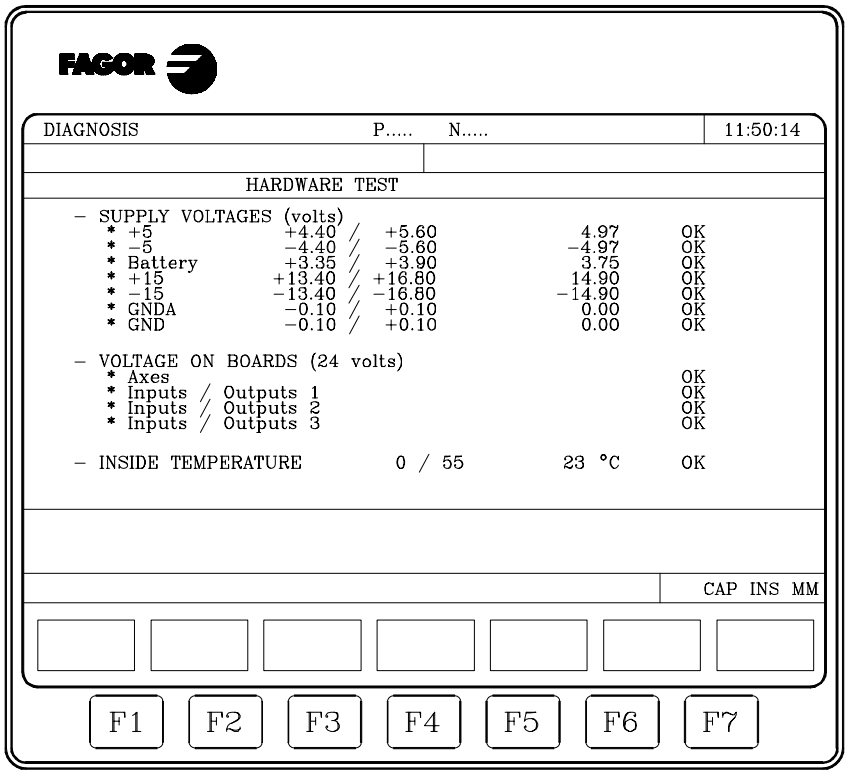
* The software version installed.
  Both for the CNC and the HD module.

* The id codes of the unit. They are only to be used by the Service Department.

The [CODE VALIDATION] softkey must be used after consulting with the Service Department when wishing to implement more software features.

12.2 HARDWARE TEST

This option checks the power supply voltages corresponding to the system and to the boards as well as the internal temperature of the central unit. It displays the following information:


FAGOR
DIAGNOSIS P..... N..... 11:50:14
HARDWARE TEST
– SUPPLY VOLTAGES (volts)
* +5 +4.40 / +5.60 4.97 OK
* –5 –4.40 / –5.60 –4.97 OK
* Battery +3.35 / +3.90 3.75 OK
* +15 +13.40 / +16.80 14.90 OK
* –15 –13.40 / –16.80 –14.90 OK
* GNDA –0.10 / +0.10 0.00 OK
* GND –0.10 / +0.10 0.00 OK
– VOLTAGE ON BOARDS (24 volts)
* Axes OK
* Inputs / Outputs 1 OK
* Inputs / Outputs 2 OK
* Inputs / Outputs 3 OK
– INSIDE TEMPERATURE 0 / 55 23 °C OK
CAP INS MM
F1 F2 F3 F4 F5 F6 F7

SUPPLY VOLTAGE

It indicates the voltage of the lithium battery and the voltages supplied by the Power Supply Module.

The voltages supplied by the Power Supply module are internally used by the CNC.

Next to the voltages, it displays the value range (maximum and minimum values), the real value and whether it is OK or not.

BOARD VOLTAGE

This section indicates whether the AXES module, the I/O TRACING module and the I/O modules are supplied or not with 24 V.

The lack of these 24V may be because the connectors have not been supplied or because the protection fuse for the corresponding module is blown.

INSIDE TEMPERATURE

It shows the value range (maximum and minimum values), the inside temperature of the Central Unit and whether that value is OK or not.

12.3 MEMORY TEST

This option checks the status of the internal CNC memory, that of the memory available for the User and for the System.

To carry out this verification, the PLC program must be stopped, otherwise, the CNC will ask the operator whether this operation is to be carried out or not.

12.4 FLASH MEMORY TEST

This option checks the status of the internal CNC Flash memory. These memories contain the CNC software version currently installed.

12.5 USER

This option will execute the program which is selected with the general machine parameter "**USERDIAG**" in the user channel.

To quit its execution and return to the previous menu, press **ESC**

12.6 HARD DISK

Once this option has been selected, two softkeys will be displayed:

Test	It check the status of the hard disk (user memory available). It takes about 30 minutes. In order to perform this test, the PLC program must be stopped. If it is running, the CNC will ask the operator whether it is to be stopped or not.
Compress	It compresses the hard disk by defragmenting it. It also includes a hard disk surface check. The duration of this test depends on the number of files it contains and on how defragmented the hard disk is.

12.7 INTERESTING NOTES

The **CNC** carries out a series of sequential tests.

If the result obtained is not correct, it may stop axes feed and spindle rotation (by cancelling their analog voltages and Enables), as well as stopping the execution of the **PLC** program or activating the external **EMERGENCY** output (**01**).

Test type	**When is it carried out?**	**Stops the axes and the spindle**	**Stops the PLC**	**Activates Emergency output**
Temperature	Always	YES	No	YES
Battery out	Always	No	No	No
Flash memory	From Diagnosis	YES	YES	V
(CARD A)	On power-up	No	No	No
RAM memory	FromDiagnosis	YES	No	YES
External emergency	EXEC/SIMUL	YES	No	YES
Board voltage	EXEC/SIMUL	YES	No	YES
PLC running	EXEC/SIMUL	YES	---	YES
PLC user error	EXEC/SIMUL	YES	No	No
PLC Watchdog	PLC running	YES	YES	YES

CNC 8055 T

Programming Manual

Ref. 0001 (in)

Please note that some of the features described in this manual might not be implemented in the software version that you just obtained.

Probing canned cycles	Option
Tool life monitoring	Option
DNC	Option
Software for 4 axes	Option
Software for 7 axes	Option
Profile editor	Option
"C" axis (Lathe)	Option
Tangential control	Option
Conversational software (TC and TCO)	Option

---------- o ----------

The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

INDEX

VERSION HISTOTY

INTRODUCTION

1. OVERVIEW

1.1 Part-programs 1
1.1.1 Considerations for the Ethernet connection 4
1.2 DNC connection 6
1.3 Communication protocol via DNC or peripheral device 6

2. CREATING A PROGRAM

2.1 Creating a program in the CNC 1
2.1.1 Block header 2
2.1.2 Program block 3
2.1.2.1 ISO language 3
2.1.2.2 High level language 3
2.1.3 End of block 4

3. AXES AND COORDINATE SYSTEMS

3.1 Nomenclature of the axes 1
3.1.1 Selection of the axes 2
3.2 Plane selection (G16, G17, G18, G19) 3
3.3 Dimensioning part millimeters (G71) / inches (G70) 5
3.4 Absolute/incremental programming (G90, G91) 6
3.5 Programming of coordinates 7
3.5.1 Cartesian coordinates 7
3.5.2 Polar coordinates 8
3.5.3 Angle and one cartesian coordinate 11
3.6 Rotary axes 12
3.7 Work zones 13
3.7.1 Definition of the work zones 13
3.7.2 Using work zones 14

4. REFERENCE SYSTEMS

4.1 Reference points 1
4.2 Machine reference search (G74) 2
4.3 Programming with respect to machine zero (G53) 3
4.4 Presetting of coordinates and zero offsets 4
4.4.1 Coordinate preset and limitation of the S value (G92) 6
4.4.2 Zero offsets (G54..G59) 7
4.5 Polar origin preset (G93) 9

5. PROGRAMMING BY ISO CODE

5.1 Preparatory functions 2
5.2 Feedrate F 4
5.2.1 Feedrate in mm/min or inches/min (G94) 4
5.2.2 Feedrate in mm/rev.or inches/rev (G95) 5
5.3 Spindle speed and spindle orientation (S) 6
5.3.1 Constant surface speed (G96) 7
5.3.2 Spindle rotation speed in rpm (G97) 7
5.4 Spindle selection (G28, G29) 8
5.5 Synchronized spindles (G30, G77S, G78S) 9
5.6 Tool number (T) and tool offset (D) 10
5.7 Miscellaneous function (M) 12
5.7.1 M00. Program stop 13
5.7.2 M01. Conditional program stop 13
5.7.3 M02. End of program 13
5.7.4 M30. End of program with return to first block 13
5.7.5 M03. Clockwise spindle rotation 13
5.7.6 M04. Counterclockwise spindle rotation 13
5.7.7 M05. Spindle stop 13
5.7.8 M06. Tool change 14
5.7.9 M19. Spindle orientation 14
5.7.10 M41, M42, M43, M44. Spindle speed range change 15
5.7.11 M45 Auxiliary spindle / live tool 15

6. PATH CONTROL

6.1 Rapid travel (G00) 2
6.2 Linear interpolation (G01) 3
6.3 Circular interpolation (G02. G03) 4
6.4 Circular interpolation by programming the center of the arc in absolute coordinates (G06) 9
6.5 Arc tangent to the previous path (G08) 10
6.6 Arc defined by three points (G09) 11
6.7 Helical interpolation 12
6.8 Tangential entry at the beginning of a machining operation (G37) 13
6.9 Tangential exit at the end of a machining operation (G38) 15
6.10 Automatic radius blend (G36) 17
6.11 Automatic chamfer blend (G39) 19
6.12 Electronic thread cutting (G33) 20
6.13 Activate "C" axis (G15) 23
6.13.1 Machining on a cylindrical surface 23
6.13.2 Face machining 25
6.14 Move to hardstop (G52) 26
6.15 Feedrate "F" as an inverted function of time (G32) 27
6.16 Tangential control (G45) 28
6.16.1 Considerations about function G45 30

7. ADDITIONAL PREPARATORY FUNCTIONS

7.1 Interruption of block preparation (G04) 1
7.2 Dwell (G04 K) 3
7.3 Working with square (G07) and round (G05,G50)corners 4
7.3.1 Square corner (G07) 4
7.3.2 Round corner (G05) 5
7.3.3 Controlled round corner (G50) 6
7.4 Look-ahead (G51) 7
7.5 Mirror image (G10, G11. G12, G13, G14) 9
7.6 Scaling factor (G72) 10
7.6.1 Scaling factor applied to all axes 11
7.6.2 Scaling factor applied to one or more axes 12
7.7 Slaved axis/cancellation of slaved axis 15
7.7.1 Slaved axis (G77) 16
7.7.2 Slaved axis cancellation (G78) 17

8. TOOL COMPENSATION

8.1 Tool radius compensation (G40, G41, G42) 4
8.2 Activating tool radius compensation 5
8.3 Tool radius compensation sections 8
8.4 Cancelling tool radius compensation 11
8.5 Temporary tool radius cancellation with G00 14
8.6 Tool compensation in any plane 16

9. CANNED CYCLES

9.1 G66. Pattern repeat canned cycle 2
9.2 G68. Stock removal cycle along X axis 9
9.3 G69. Stock removal canned cycle along Z axis 17
9.4 G81. Turning canned cycle with straight sections 26
9.5 G82. Facing canned cycle with straight sections 30
9.6 G83. Axial drilling and tapping canned cycle 34
9.7 G84. Turning canned cycle with arcs 36
9.8 G85. Facing canned cycle with arcs 40
9.9 G86. Longitudinal threadcutting canned cycle 44
9.10 G87. Face threadcutting canned cycle 49
9.11 G88. Grooving canned cycle along the x axis 54
9.12 G89. Grooving canned cycle along the Z axis 56
9.13 G60. Axial drilling / tapping 58
9.14 G61. Radial drilling/ tapping 61
9.15 G62. Longitudinal slot milling cycle 65
9.16 G63. Radial slot milling cycle 67

10. PROBING

10.1 Probing (G75,G76) 1
10.2 Probing canned cycles 2
10.3 Tool calibration canned cycle 3
10.4 Probe calibration canned cycle 9
10.5 Part measuring and tool offset correction canned cycle along X axis 13
10.6 Part measuring and tool offset correction canned cycle along Z axis 16

11.PROGRAMMING IN HIGH-LEVEL LANGUAGE

11.1 Lexical description 1
11.1.1 Reserved words 2
11.1.2 Numerical constants 3
11.1.3 Symbols 3
11.2 Variables 4
11.2.1 General purpose parameters or variables 6
11.2.2 Variables associated with tools 8
11.2.3 Variables associated with zero offsets 10
11.2.4 Variables associated with machine parameters 12
11.2.5 Variables associated with work zones 13
11.2.6 Variables associated with feedrates 14
11.2.7 Variables associated with coordinates 16
11.2.8 Variables associated with the electronic handwheels 17
11.2.9 Variables associated with the main spindle 19
11.2.10 Variables associated with the second spindle 22
11.2.11 Variables asociadas a herramienta motorizada 24
11.2.12 Variables associated with the plc 25
11.2.13 Variables associated with local parameters 27
11.2.14 Sercos variables 28
11.2.15 Other variables 29
11.3 Constants 35
11.4 Operators 35
11.5 Expressions 37
11.5.1 Arithmetic expressions 37
11.5.2 Relational expressions 38

12. PROGRAM CONTROL STATEMENTS

12.1 Assignment statements 1
12.2 Display statements 2
12.3 Enabling-disabling statements 3
12.4 Flow control statements 4
12.5 Subroutine statements 6
12.5.1 Interruption subroutine statements 11
12.6 Program statements 12
12.7 Screen customizing statements (graphic editor) 15

APPENDIX

ISO code programming 3
Internal CNC variables 5
High-level programming 11
Key codes 13
Logic outputs for key codes status 14
Keys inhibiting codes 15
Programming assistance system pages 16
Maintenance 19

VERSION HISTORY (T)

(LATHE MODEL)

Date: **December 1999** ***Software Version:*** **4.0x**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Portuguese language	Installation manual	Chap. 3
Tangential control	Installation manual Programming manual	Chap. 9, Chap. 10, Appendix Chap. 6, Chap. 11, Appendix
PLC. user registers from R1 to R499	Installation manual Programming manual	Chap. 6, Chap. 7, Appendix Chap. 11
CNC status screen	Operating manual	Chap. 8
Hard Disk (HD)	Installation manual	Chap. 1, Chap. 3, Appendix
Diagnosis of the HD	Operating manual	Chap. 12,
Integrate the HD in an external PC network	Installation manual	Chap. 3
Consult directories, delete, rename and copy programs in the same or another device.	Operating manual Programming manual	Chap. 1, Chap. 7 Chap. 1
Execution and simulation from RAM, Memkey Card, HD or serial line.	Operating manual	Chap. 1, Chap. 3,
It is now possible to execute (EXEC) and open (OPEN) for editing a program stored in any device.	Programming manual	Chap. 14, Appendix
Thread repair. Reference (home) the spindle before.	Programming manual TC operating manual	Chap. 9 Chap. 4
Simulation in rapid, without assuming G95 or M3, M54, etc.	Operating manual	Chap. 3
Geometry associated with the tool offset.	Installation manual Operating manual	Chap. 3 Chap. 6
Live tool with M45 or as if it were a 2nd spindle	Installation manual	Chap. 3
PLC channel affected by another feedrate override set by PLC.	Installation manual	Chap. 11
Independent x1, x10, x100 factor for each handwheel.	Installation manual Programming manual	Chap. 4, Chap. 10, Appendix Chap. 11
Handling the Fagor HBE handwheel	Installation manual	Chap. 4, 9, 10, Appendix
Spindle synchronization (G77 S)	Installation manual Programming manual	Chap. 3, 9, 10, Appendix Chap. 5, 11, Appendix
Optimizing of profile machining.	Programming manual TC operating manual	Chap. 9 Chap. 4
(2) axes controlled by a single servo drive	Installation manual	Chap. 3, 4, 9, Appendix
G75 function affected by Feedrate override (%)	Installation manual	Chap. 3
Probe. Probe position by cycle parameters.	Programming manual	Chap. 10
Protection against deleting OEM screens	Operating manual	Chap. 7
Detecting temperature and battery voltage on the new CPU.		
TC option. ISO program management, also like MDI.	TC operating manual	Chap. 3
TC option. Coolant icon in all cycles.	TC operating manual	Chap. 4
TC option. Background editing.	TC operating manual	Chap. 4
TC option. Key codes for user cycles.	TC operating manual	Appendix

INTRODUCTION

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Protections of the unit itself

Power Supply Module

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input

Axes module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against reverse connection of the power supply.

Input / Output Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Input / Output and Tracing Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Fan Module

It carries 1 or 2 external fuses depending on model
The fuses are fast (F), of 0.4 Amp./ 250V. to protect the fans.

Monitor

The type of protection fuse depends on the type of monitor. See the identification label of the unit itself.

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

Safety symbols

Symbols which may appear on the manual

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

"Electrical Shock" symbol
It indicates that point may be under electrical voltage

"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the Monitor or the Central Unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

When sending the monitor, especially protect the CRT glass

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE CNC

OEM Manual	Is directed to the machine builder or person in charge of installing and starting-up the CNC.
USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual describing how to operate the CNC. Programming Manual describing how to program the CNC.
DNC Software Manual	Is directed to people using the optional DNC communications software.
DNC Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the CNC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The Programming Manual for the Lathe model CNC contains the following chapters:

Index

New Features and Modifications for the Lathe Model

Introduction
- Summary of safety conditions
- Shipping terms
- Fagor documentation for the CNC.
- Manual contents

Chapter 1 Overview
- It shows how to enter part-programs from the keyboard or via DNC.
- Location of part-programs, how to edit and execute them.
- It indicates the protocol to be used in DNC communications.

Chapter 2 Creating a program
- It indicates the structure for a part-program and all its blocks.
- It shows the languages that could be used to program the parts: ISO coded and High-Level languages

Chapter 3 Axes and coordinate systems
- It indicates the nomenclature of the axes and how to select them.
- It shows how to select the working planes, work units, type of programming system (absolute /incremental).
- It describes the coordinates systems that could be used for programming: Cartesian, polar, cylindric, angle plus Cartesian coordinate.
- It shows how to operate with rotary axes and how to define and use work zones.

Chapter 4 Reference systems
- It indicates the machine reference (home) and datum points to be set at the CNC.
- It shows how to program a home search, how to program coordinates with respect to home, how to preset coordinates, zero offsets and polar origins.

Chapter 5 Programming by ISO code
- It shows how to program preparatory functions for feedrate and constant speed as well as additional functions such as “F, S, T, D and M”.

Chapter 6 Path control
- It shows how to program rapid traverse, linear, circular and helical interpolations.
- It shows how to program tangential entries and exits as well as corner rounding and chamfering
- It shows how to program electronic threading and movements against hard stop.
- It shows how to program the “C” axis to machine along the surface and on the face of the part.
- It shows how to program the tangential control.

Chapter 7 Additional preparatory functions
- It shows how to interrupt block preparation and how to program a dwell.
- It shows how to program a part in square corner, round corner or with an automatic radius blend.
- It describes how to program the look-ahead, mirror image, scaling factor, pattern rotation and the electronic slaving / unslaving of the axes.

Chapter 8 Tool compensation
- It shows how to program tool radius and length compensation.

Chapter 9 Canned cycles
- It shows how to program the different machining canned cycles.

Chapter 10 Working with a probe
- It shows how to carry out probing moves and how to program the probing canned cycles.

Chapter 11 Programming in high level language

It shows all the variables, symbols, operators, etc. to be used when programming in high level language.

Chapter 12 Program control statements

It shows the control sequences that can be used in high-level language. The available instructions are: for assignment, display enable/disable, flow control, subroutines and for generating programs and screens.

Appendix
- A ISO code programming
- B Internal CNC variables
- C High level programming
- D Key codes
- E Programming assistance system pages

1. OVERVIEW

The CNC can be programmed both at the machine (from the front panel) or from external peripheral devices (tape reader/cassette recorder, computer, etc. Memory available to the user for carrying out the part programs is 1 Mbyte.

The part programs and the values in the tables which the CNC has can be entered as follows :

* **From the front panel**. Once the editing mode or table required has been selected, the CNC allows you to enter data from the keyboard.

* **From a Computer (DNC) or Peripheral Device**. The CNC allows data to be interchanged with a computer or peripheral device, using RS232C and RS422 cables.

  If this is controlled from the CNC, it is necessary to preset the corresponding table or part program directory (utilities) you want to communicate with.

  Depending on the type of communication required, the serial port machine parameter "PROTOCOL" should be selected.

  "PROTOCOL" = 0 if the communication is with a peripheral device.
  "PROTOCOL" = 1 if the communication is via DNC.

1.1 PART-PROGRAMS

Editing

To create a part-program, access the Edit mode. See chapter 5 in this manual.

The new part-program edited is stored in the CNC's RAM memory.

A copy of the part-programs may be stored in the "MemKey Card", at a PC connected through serial line 1 or 2 or in the hard disk (HD module). See chapter 7 in this manual.

When using a PC through serial line 1 or 2, proceed as follows:
- Execute the "Fagor50.exe" applications program at the PC.
- Activate DNC communications at the CNC. See chapter 8 in this manual.
- Select the work directory as shown in chapter 7 of this manual. Option: Utilities\ Directory\ Serial L.\ Change directory.

With the Edit mode of operation, part-programs residing in the CNC's RAM memory may be modified. To modify a program stored in the "MemKey Card", in a PC or in the hard disk, it must be previously copied into RAM memory.

Execution

Part-programs stored anywhere may be executed or simulated. See chapter 3 in this manual.

The user customizing programs must be in RAM memory so the CNC can execute them.

The GOTO and RPT instructions cannot be used in programs executed from a PC connected through the serial lines. See chapter 14 of the programming manual.

The subroutines can only be executed if they reside in the CNC's RAM memory. Therefore, to execute a subroutine stored in the "MemKey Card", in a PC or in the hard disk, it must be first copied into the CNC's RAM memory.

From a program in execution, another program can be executed which is in RAM memory, in the "MemKey Card", in a PC or in the hard disk using the EXEC instruction. See chapter 14 of the programming manual.

Utilities

This operating mode, chapter 7 of this manual, lets display the part-program directory of all the devices, make copies, delete, rename and even set the protections for any of them.

Ethernet

When having the Ethernet option and if the CNC is configured as another node within the computer network, the following operations are possible from any PC of the network:

- Access the part-program directory of the Hard Disk(HD).
- Edit, modify, delete, rename, etc.the programs stored on the hard disk (HD).
- Copy programs from the hard disk to the PC and vice versa.

To configure the CNC as another node within the computer network, see section 3.3.4 of the installation manual.

Operations that may be carried out with part-programs:

	RAM Memory	CARD A	HD	DNC
Consult the program directory in ...	Yes	Yes	Yes	Yes
Consult the subroutine directory in ...	Yes	No	No	No
Create work directory in ..	No	No	No	No
Change work directory in ..	No	No	No	Yes
Edit a program in ..	Yes	No	No	No
Modify a program in ..	Yes	No	No	No
Delete a program from ..	Yes	Yes	Yes	Yes
Copy from/to RAM memory to/from ...	Yes	Yes	Yes	Yes
Copy from/to CARD A to/from ...	Yes	Yes	Yes	Yes
Copy from/to HD to/from ...	Yes	Yes	Yes	Yes
Copy from/to DNC to/from ...	Yes	Yes	Yes	Yes
Rename a program in ..	Yes	Yes	Yes	No
Change the comment of a program in ..	Yes	Yes	Yes	No
Change protections of a program in ..	Yes	Yes	Yes	No
Execute a part-program in ..	Yes	Yes	Yes	Yes
Execute a user program in ..	Yes	No	No	No
Execute the PLC program in ..	Yes	*	No	No
Execute programs using the GOTO or RPT instructions from ..	Yes	Yes	Yes	No
Execute subroutines stored in ..	Yes	No	No	No
Execute programs stored in RAM, CARD A or HD using the EXEC instruction from ..	Yes	Yes	Yes	Yes
Execute programs via DNC with the EXEC instruction from ..	Yes	Yes	Yes	No
Open programs stored in RAM, CARD A or HD using the OPEN instruction from ..	Yes	Yes	Yes	Yes
Open programs via DNC using the OPEN instruction from ..	Yes	Yes	Yes	No
Consult from a PC and through Ethernet, the program directory in ...	No	No	Yes	No
Consult from a PC and through Ethernet, the subroutine directory in ...	No	No	No	No
Create from a PC and through Ethernet, a directory in...	No	No	No	No

* If it is not in RAM memory, it generates an executable code in RAM and it executes it..

1.1.1 CONSIDERATIONS FOR THE ETHERNET CONNECTION

When configuring the CNC as another node in the computer network , the programs stored in the hard disk module (HD) may be edited and modified from any PC.

Instructions for seting up a user PC to access CNC directories

Recommended configuration:

- Open the «Windows Explorer»
- On the «Tools» menu, select the «Connect to Network Drives» option.
- Select the Drive. For example: «D»
- Indicate the path: CNC name followed by the name of the shared directory. For example: \\FAGOR8055\CNCHD
- When selecting the option: «Connect again when initiating the session», the selected CNC will appear on each power-up as another path of the «Windows Explorer» without having to define it again.

This connection is established through Ethernet and, therefore, the CNC does not control the syntax of the programs while they are received or modified.

Whenever accessing the program directory of the Hard Disk (HD), the following verification takes place:

File name.

The file number must always have 6 digits and the extension PIM (for milling) or PIT (for lathe). Examples: 001204.PIM 000100.PIT

If the file has been given the wrong name, for example: 1204.PIM or 100.PIT, the CNC will not change it, but it will display it with the comment: '**********'.

The file cannot be modified from the CNC. It must be edited from the PC to correct the error.

File size.

If the file is empty, size = 0, the CNC will display it with the comment: '**********'.

The file can be edited or deleted either from the CNC or from the PC.

First line of the program

The first line of the program must have the % character, the comment associated with the file (up to 20 characters) and between the 2 commas (,) the program attributes: O (OEM), H (hidden), M (modifiable), X (executable).

Examples %Comment ,MX,
% ,OMX,

If the first line does not exist. The CNC will display it with an empty comment and with the modifiable (M) and executable (X) attributes.

When the format of the first line is wrong, the CNC does not modify it, but it displays it with the comment: '**********'. The file can be modified or deleted from the CNC or from the PC.

It is the wrong format when:
- the comment has more than 20 characters
- a comma (,) is missing for separating the attributes
- the attributes have a strange character

1.2 DNC CONNECTION

The CNC offers as optional feature the possibility of working in DNC (Distributed Numerical Control), enabling communication between the CNC and a computer to carry out the following functions :

* Directory and delete commands.
* Transfer of programs and tables between the CNC and a computer.
* Remote control of the machine.
* The ability to supervise the status of advanced DNC systems.

1.3 COMMUNICATION PROTOCOL VIA DNC OR PERIPHERAL DEVICE

This type of communication enables program-and-table transfer commands, plus the organization of CNC directories such as the Computer Directory, for copying/deleting programs, etc. to be done either from the CNC or the computer.

When you want to transfer files, it is necessary to follow this protocol :

* The "%" symbol will be used to start the file, followed by the **program comment** (optional), of up to 20 characters.

  Then, and separated by a comma ",", comes the attribute (protection) each file has: reading, modifying, etc. This protection is optional and does not have to programmed.

  To end the file header, **RETURN (RT)** or **LINE FEED (LF)** characters should be sent separated by a comma (",").

  Example :

  ```
  %Fagor Automation, -MX, RT
  ```

* Following the header, the file blocks should be programmed. These will all be programmed according to the programming rules indicated in this manual. After each block, to separate it from the others, the **RETURN (RT)** or **LINE FEED (LF)** characters should be used.

  Example :

  ```
  N20 G90 G01 X100 Y200 F2000 LF
  (RPT N10, N20) N3 LF
  ```

If communication is made with a peripheral device, you will need to send the 'end of file' command. This command is selected via the machine parameter for the serial port: "EOFCHR", and can be one of the following characters :

ESC ESCAPE
EOT END OF TRANSMISSION
SUB SUBSTITUTE
EXT END OF TRANSMISSION.

2. *CREATING A PROGRAM*

A CNC (numerical control) program consists of a series of blocks or instructions.

These blocks or instructions are made of words composed of capital letters and numerical format.

The CNC's numerical format consists of :

- the symbols **. + -**
- the figures **0 1 2 3 4 5 6 7 8 9**

Programming allows spaces between letters, numbers and symbols, in addition to ignoring the numerical format if it has zero value, or a symbol if it is positive.

The numerical format of a word can be replaced by an arithmetic parameter in programming. Later and during basic execution, the control will replace the arithmetic parameter by its value, for example :

If XP3 has been programmed, during execution the CNC will replace P3 by its numerical value, obtaining results such as X20, X20.567, X-0.003, etc.

2.1 CREATING A PROGRAM IN THE CNC

All the blocks which make up the program have the following structure :

Block header + program block + end of block

2.1.1 BLOCK HEADER

The block header is optional, and may consist of one or more **block skip conditions** and by the **block number or label**. Both can be programmed in this order.

CONDITION FOR BLOCK SKIP, /, /1, /2, /3.

These three block skip conditions, given that "/" and "/1" is the same, are governed by the marks BLKSKIP1, BLKSKIP2 and BLKSKIP3 of the PLC.

If any of these marks is active, the CNC will not execute the block or blocks in which it has been programmed. The execution takes place in the following block.

Up to 3 skip conditions can be programmed in one block. These will be evaluated one by one, respecting the order in which they have been programmed.

The control reads 20 blocks ahead of the one being executed in order to calculate in advance the path to be run.

The condition for block skip will be analyzed at the time when the block is read i.e. 20 blocks before execution.

If the block skip needs to be analyzed at the time of execution, it is necessary to interrupt the block preparation, by programming G4 in the previous block.

BLOCK LABEL OR NUMBER N(0-9999)

This is used to identify the block, and is only used when block references or jumps are made.

They are represented by the letter N followed by up to 4 figures (0-9999). It is not necessary to follow any order, and randomly arranged numbers are allowed.

If two or more blocks with the same label number are present in the same program, the CNC will always give priority to the first number.

Although it is not necessary to program it, by using a SOFTKEY the CNC allows the automatic programming of labels. The programmer can select the initial number and the step between labels.

2.1.2 PROGRAM BLOCK

This is written with commands in ISO and High Level languages.

To prepare a program, blocks written in both languages will be used, although each one should be edited with commands in just one language.

2.1.2.1 ISO LANGUAGE

This language is specially designed to control axis movement, as it gives information and movement conditions, in addition to data on feedrate. It includes :

* Preparatory functions for movement, used to determine geometry and working conditions, such as linear and circular interpolations, threading, etc.

* Control functions for axis feedrate and spindle speeds.

* Tool control functions.

* Complementary functions, with technological instructions.

2.1.2.2 HIGH LEVEL LANGUAGE

This enables access to general purpose variables and to system tables and variables.

It gives the user a number of control sentences which are similar to the terminology used in other languages, such as : IF, GOTO, CALL, etc.

It also allows the use of any type of expression (arithmetic, referential, or logical).

It also has instructions for the construction of loops, plus subroutines with local variables. “Local variable” is understood to mean one which is only recognized by the subroutine in which it has been defined.

It is also possible to create libraries, grouping subroutines with useful and tested functions, which can be accessed from any program.

2.1.3 END OF BLOCK

The end of block is optional and may consist of the indication of **number of repetitions of the block** and of the **block comment**. Both must be programmed in this order.

NUMBER OF REPETITIONS OF THE BLOCK, N(0-9999)

This indicates the number of times the block will be executed.

Movement blocks can only be repeated which, at the time of their execution, are under the influence of a modal subroutine.

In these cases, the CNC executes the programmed move and the active machining operation (canned cycle or modal subroutine) the indicated number of times.

The number of repetitions is represented by the letter N followed by up to 4 digits (0-9999).

The active machining operation does not take place if N0 is programmed. Only the movement programmed within the block takes place.

BLOCK COMMENT

The CNC allows you to incorporate any kind of information into all blocks in the form of a comment.

The comment is programmed at the end of the block, and should begin with the character "**;**".

If a block begins with ";", all its contents will be considered as a comment, and it will not be executed.

Empty blocks are not permitted. They should contain at least one comment.

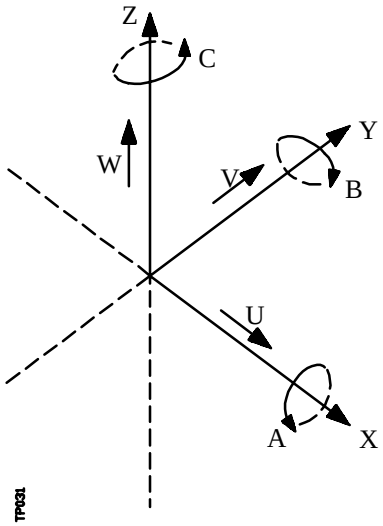
3. *AXES AND COORDINATE SYSTEMS*

Given that the objective of the CNC is to control the movement and positioning of axes, it is necessary to determine, by means of coordinates, the position of the point to be reached.7

The CNC allows you to use absolute, relative or incremental coordinates throughout the same program.

3.1 NOMENCLATURE OF THE AXES

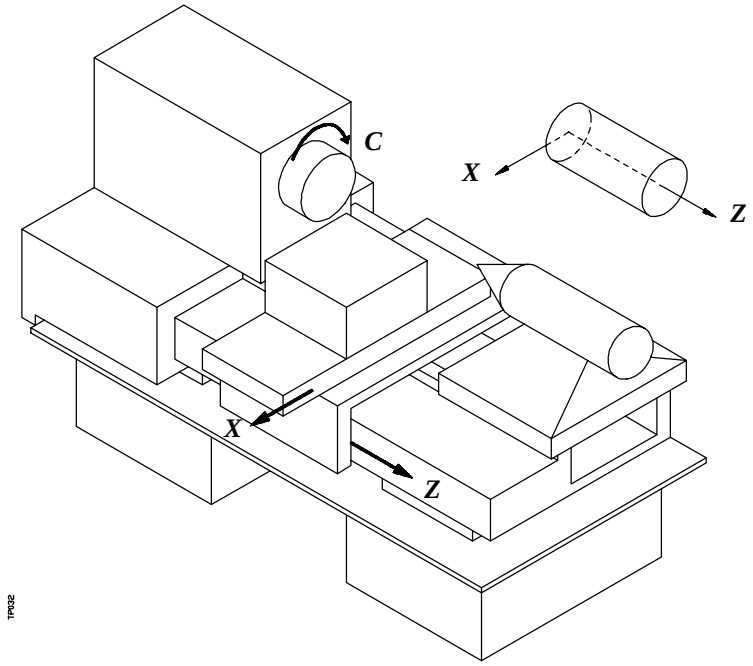
The axes are named according to DIN 66217.



Characteristics of the system of axes :

- *** X & Y** main movements on the main work plane of the machine.
- *** Z** parallel to the main axis of the machine, perpendicular to the main XY plane.
- *** U,V,W** auxiliary axes parallel to X,Y, Z respectively
- *** A,B,C** rotary axes on each of the X,Y, Z axes.

The drawing below shows an example of the nomenclature of the axes on a parallel lathe.



3.1.1 SELECTION OF THE AXES

Of the 9 possible axes which can exist, the CNC allows the manufacturer to select up to 7 of them.

Moreover, all the axes should be suitably defined as linear/rotary, etc. through the axis machine parameters which appear in the Installation and Start-up Manual.

There is no limitation to the programming of the axes, and interpolations can be made simultaneously with up to 7 axes.

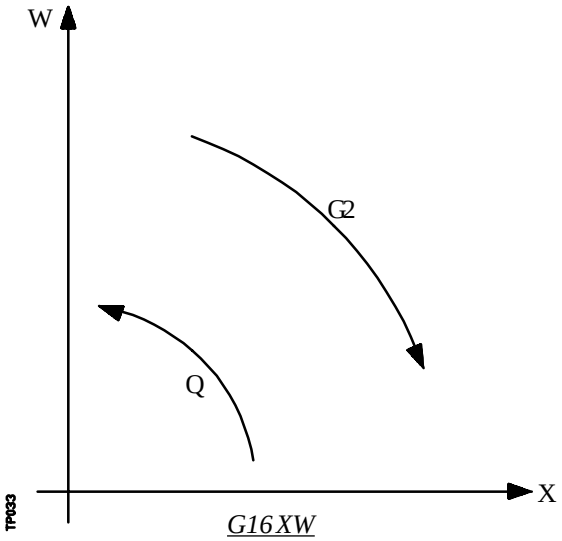
3.2 PLANE SELECTION (G16, G17, G18, G19)

Plane selection should be made when the following are carried out :

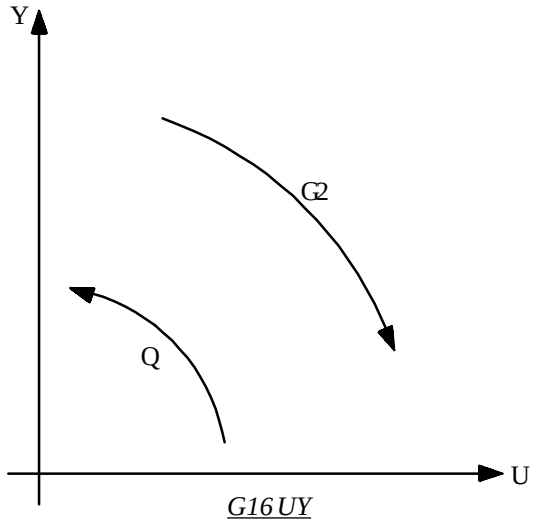
- Circular interpolations.
- Controlled corner rounding.
- Tangential entry and exit.
- Chamfer blend.
- Position values programming in polar coordinates.
- Tool radius Compensation.
- Tool length compensation.

The "G" functions which enable selection of work planes are as follows :

* **G16 axis1 axis2**. Enables selection of the desired work plane, plus the direction of G02 G03 (circular interpolation), **axis1** being programmed as the abscissa axis and **axis2** as the ordinate axis.



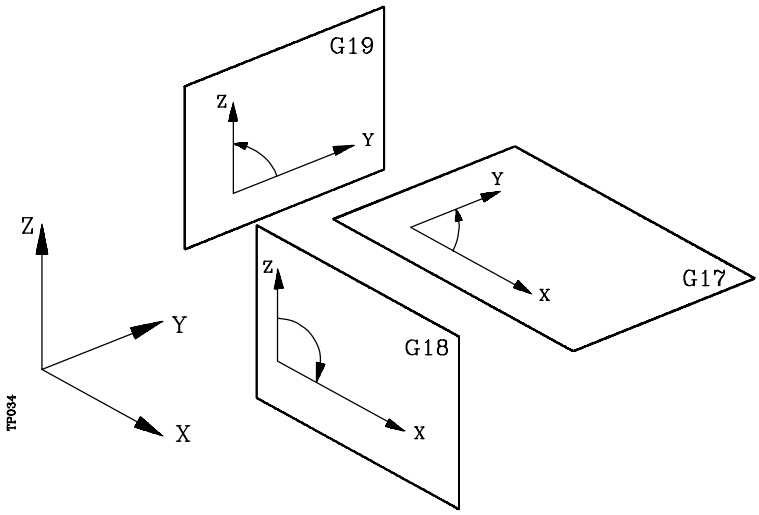
G16 XW



G16 UY

* **G17**. Selects the XY plane
* **G18**. Selects the ZX plane
* **G19**. Selects the YZ plane

The G16, G17, G18 and G19 functions are modal and incompatible among themselves. The G16 function should be programmed on its own within a block.



On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the plane defined by the general machine parameter as "IPLANE" is the work plane.

3.3 DIMENSIONING PART MILLIMETERS (G71) / INCHES (G70)

The CNC allows you to enter units of measurement with the programming, either in millimeters or inches.

It has a general machine parameter "INCHES" to define the unit of measurement of the CNC.

However, these units of measurement can be changed at any time in the program. Two functions are supplied for this purpose :

* G70 : Programming in inches.
* G71 : Programming in millimeters.

Depending on whether G70 or G71 has been programmed, the CNC assumes the corresponding set of units for all the blocks programmed from that moment on.

The G70 and G71 functions are modal and are incompatible.

The CNC allows the programming of figures from 0.0001 to 99999.9999 (with or without sign) when it works in millimeters (G71). This is called format ± 5.4, or from 0.00001 to 3937.00787 (with or without sign) if it is programmed in inches (G70). This is called format ± 4.5.

However, and to simplify the instructions, we can say that the CNC admits ± 5.5 format, thereby admitting ± 5.4 in millimeters and ± 4.5 in inches.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the system of units of measurement is the one defined by the general machine parameter "INCHES".

3.4 ABSOLUTE/INCREMENTAL PROGRAMMING (G90, G91)

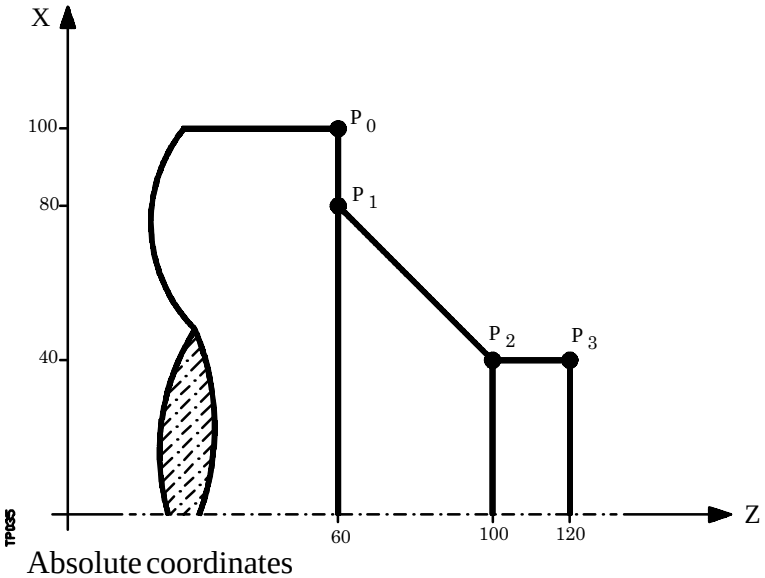
The CNC allows the programming of the coordinates of one point either with absolute **G90** or incremental **G91** values.

When working with absolute coordinates (G90), the point coordinates refer to a point of origin of established coordinates, often the part zero (datum).

When working in incremental coordinates (G91), the numerical value programmed corresponds to the movement information for the distance to be travelled from the point where the tool is situated at that time. The sign in front shows the direction of movement.

Functions G90/G91 are modal and incompatible.

Example with X axis programming in diameter:



Absolute coordinates

```
G90  X200  Z 60  ; Point P0
     X160  Z 60  ; Point P1
     X 80  Z100  ; Point P2
     X 80  Z120  ; Point P3
```

Incremental coordinates

```
G90  X200  Z 60  ; Point P0
G91  X-40        ; Point P1
     X-80  Z 40  ; Point P2
           Z 20  ; Point P3
```

On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC will assume G90 or G91 according to the definition by the general machine parameter "ISYSTEM".

3.5 PROGRAMMING OF COORDINATES

The CNC allows the selection of up to 7 from the 9 possible axes X,Y,Z,U,V,W,A,B,C.

Each of these may be linear, linear to position only, normal rotary, rotary to position only or rotary with hirth toothing (positioning in complete degrees), according to the specification in the machine parameter of each "AXISTYPE" axis.

With the aim of always selecting the most suitable coordinate programming system, the CNC has the following types :

* Cartesian coordinates
* Polar coordinates
* Angle and one Cartesian coordinate.

3.5.1 CARTESIAN COORDINATES

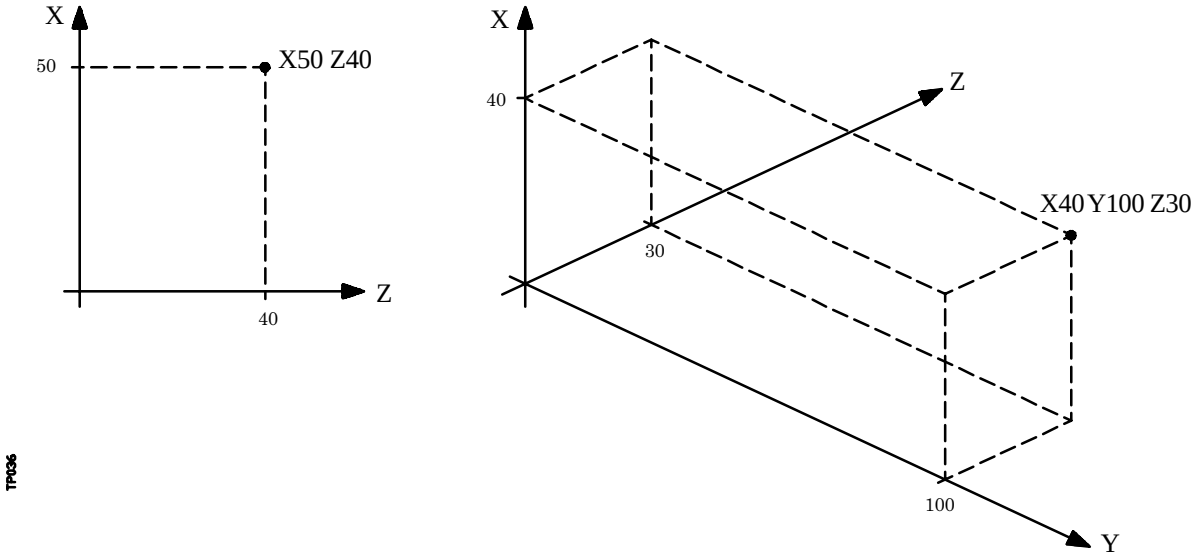
The Cartesian Coordinate System is defined by two axes on the plane, and by three or more axes in space.

The origin of all these, which in the case of the axes X Y Z coincides with the point of intersection, is called **Cartesian Origin** or **Zero Point of the Coordinate System**.

The position of the different points of the machine is expressed in terms of the coordinates of the axes, with two, three, four, or five coordinates.

The coordinates of the axes are programmed via the letter of the axis (X,Y,Z,U,V,W,A,B,C, always in this order) followed by the coordinate value.

The values of the coordinates are absolute or incremental, depending on whether it is working in G90 or G91, and its programming format is ± 5.5.

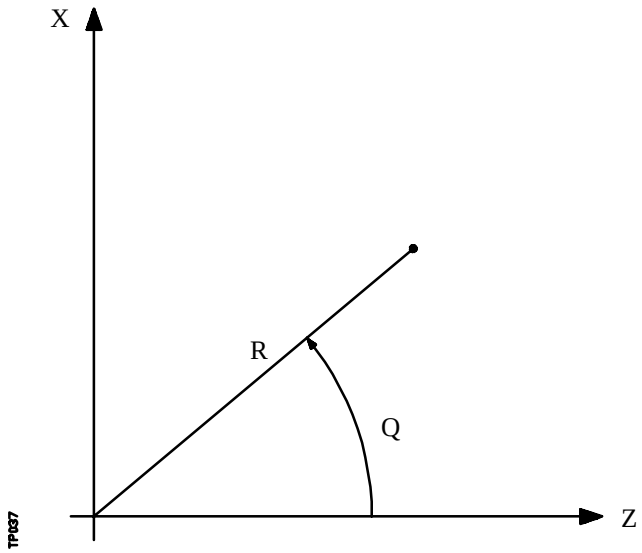


3.5.2 POLAR COORDINATES

In the event of the presence of circular elements or angular dimensions, the coordinates of the different points on the plane (2 axes at the same time), it may be easier to express them in polar coordinates.

The reference point is called **Polar Origin**, and this will be the origin of the **Polar Coordinate System**.

A point on this system would be defined by :



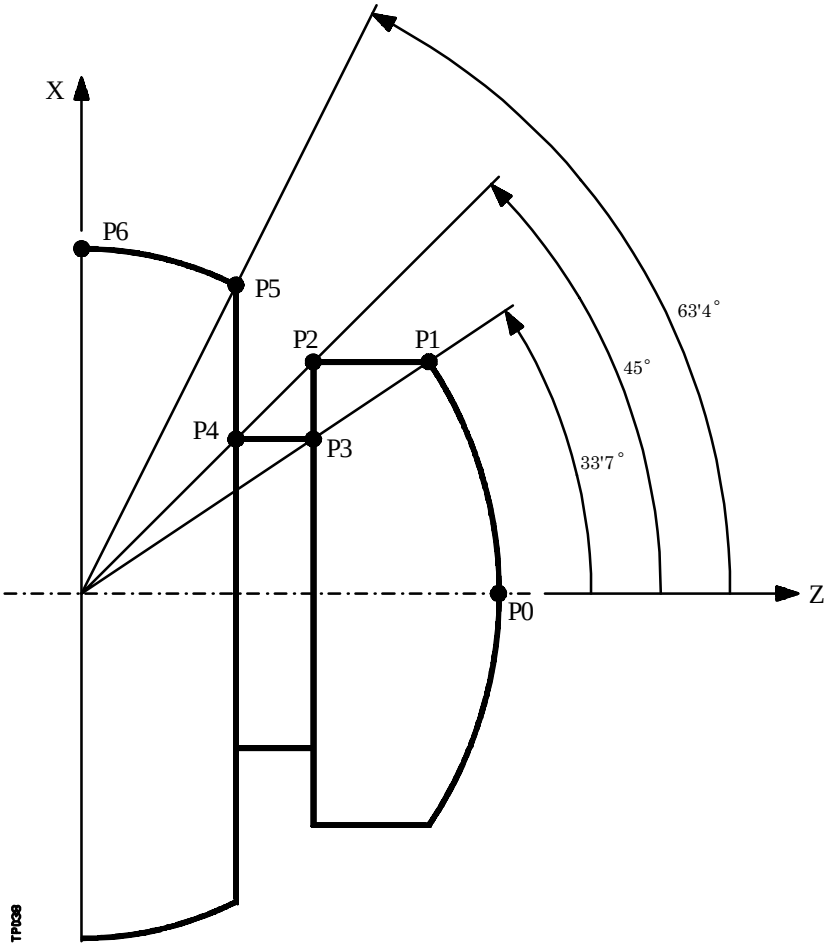
- The **RADIUS (R)**, the distance between the polar origin and the point.
- The **ANGLE (Q)**, formed by the abscissa axis and the line which joins the polar origin with the point (in degrees).

The values **R** and **Q** are absolute or incremental, depending on whether you are working with G90 or G91, and their programming format will be R ±5.5 Q±5.5. **No negative** values can be given to the radius, it must always be positive.

The programming of negative **R** values is permitted when programming in incremental coordinates, but always provided that **the resulting value assigned to the radius is positive.**

If a **Q** value over 3600 is programmed, the module will be taken after dividing it by 360. Thus, Q420 is the same a Q60, and Q-240 is the same as Q-60.

Programming example assuming that the Polar Origin is located at the Coordinate Origin.



Absolute coordinates

```
G90        R430  Q0      ; Point P0
     G03         Q33.7   ; Point P1, in an arc (G03)
     G01   R340  Q45     ; Point P2, in a straight line (G01)
     G01   R290  Q33.7   ; Point P3, in a straight line (G01)
     G01   R230  Q45     ; Point P4, in a straight line (G01)
     G01   R360  Q63.4   ; Point P5, in a straight line (G01)
     G03         Q90     ; Point P6, in an arc (G03)
```

Incremental coordinates

```
G90        R430  Q0      ; Point P0
G91  G03         Q33.7   ; Point P1, in an arc (G03)
     G01   R-90  Q11.3   ; Point P2, in a straight line (G01)
     G01   R-50  Q-11.3  ; Point P3, in a straight line (G01)
     G01   R-60  Q11.3   ; Point P4, in a straight line (G01)
     G01   R130  Q18.4   ; Point P5, in a straight line (G01)
     G03         Q26.6   ; Point P6, in an arc (G03)
```

The polar origin, apart from being able to be preset using function G93 (described later) can be modified in the following cases :

* On power-up, after executing M02, M30 EMERGENCY or RESET, the CNC will assume, as the polar origin, the coordinate origin of the work plane defined by the general machine parameter"IPLANE".

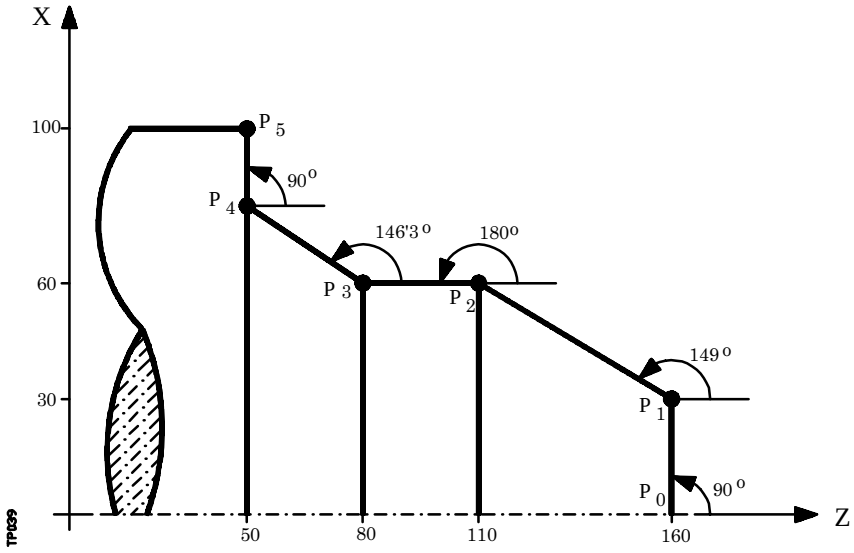
* Every time the work plane is changed (G16,G17,G18 or G19), the CNC assumes the coordinate origin of the new work plane selected as the polar origin.

* When executing a circular interpolation (G02 or G03), and if the general machine parameter "PORGMOVE" has a value of 1, the center of the arc will become the new polar origin.

3.5.3 ANGLE AND ONE CARTESIAN COORDINATE

A point on the main plane can be defined via one of its cartesian coordinates, and the exit angle of the previous path.

Example with X axis programming in radius:




```
X0      Z160  ; Point P0
Q90     X30   ; Point P1
Q149    Z110  ; Point P2
Q180    X80   ; Point P3
Q146.3  Z50   ; Point P4
Q90     X100  ; Point P5
```

If you wish to represent a point in space, the remaining coordinates can be programmed in cartesian coordinates.

3.6 ROTARY AXES

The types of rotary axes available are:

Normal rotary axis.
Positioning-only rotary axis.
Hirth rotary axis.

Each one of them can be divided into:
Rollover When their position reading goes from 0º to 360º.
No rollover When their position reading goes from -99999º to 99999º

They are all programmed in degrees. Therefore, their readings are not affected by the inch/mm conversion.

Normal rotary axes

They can be interpolated with linear axes.
Movement: in G00 and G01
Rollover axis programming:
G90 The sign indicates the turning direction and the target position (between 0 and 359.9999).
G91 The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will rotate more than one turn before positioning at the desired point.
Non-rollover axis programming: In G90 and G91 like a linear axis.

Positioning-only Axes
They cannot be interpolated with linear axes.
Movement: Always in G00 and they do not admit tool radius compensation (G41, G42).
Rollover axis programming:
G90 Always positive and via the shortest path. End coordinate between 0 & 359.9999
G91 The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will rotate more than one turn before positioning at the desired point.
Non-rollover axis programming: In G90 and G91 like a linear axis.

HIRTH axes

They work like the positioning-only axis except that they do not admit decimal position values (coordinates).

More than one hirth axis can be used, but they can only be moved one at a time.

3.7 WORK ZONES

The CNC provides four work zones or areas, and also limits the tool movement in each of these.

3.7.1 DEFINITION OF THE WORK ZONES

Within each work zone, the CNC allows you to limit the movement of the tool in each axes, with upper and lower limits being defined in each axis.

G20: Defines the lower limits in the desired zone.
G21: Defines the upper limits in the desired zone.

The format to program these functions is:

G20 K X...C +/- 5.5
G20 K X...C +/- 5.5

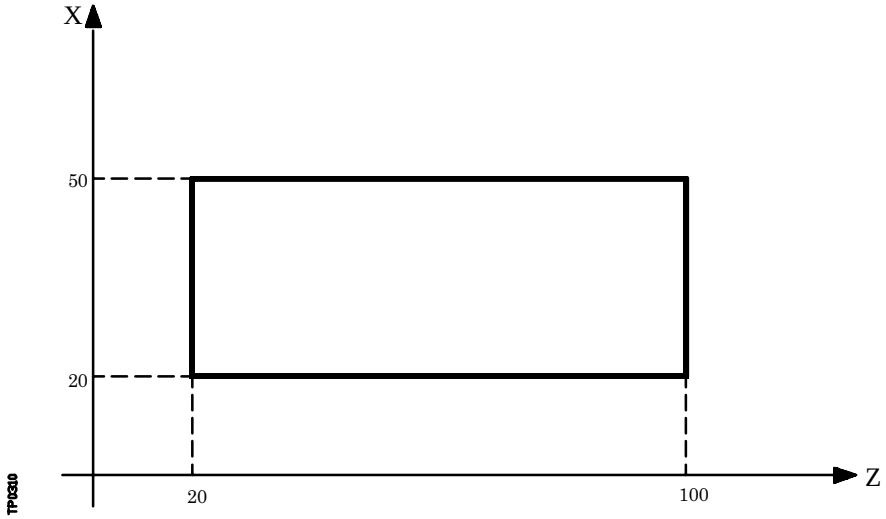
In which :

*** K** Indicates the work zone you wish to define (1, 2, 3 or 4)

*** X...C** Indicates the coordinates (upper or lower) with which you wish to limit the axes. These coordinates will be given in radius and will be programmed with reference to machine zero (home).

It is not necessary to program all the axes, so only defined axes will be limited.

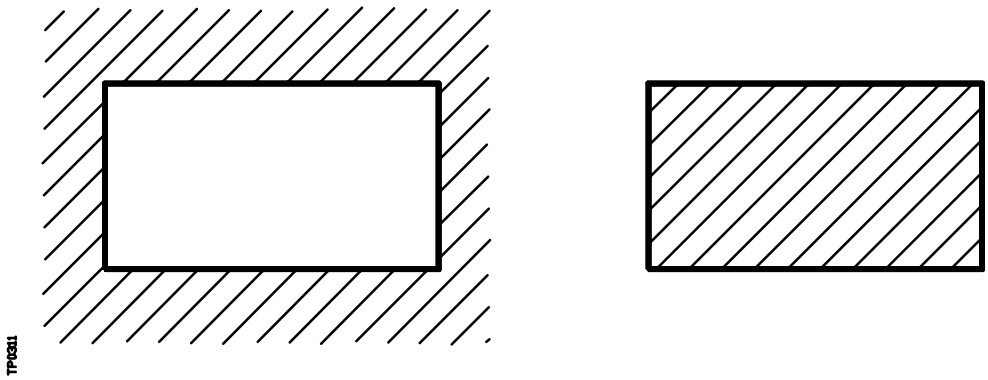
Example :



G20 K1 X20 Z20
G21 K1 X50 Z100

3.7.2 USING WORK ZONES

Within each work zone, the CNC allows you to restrict the movement of the tool, either prohibiting its exit from the programmed zone (no exit zone) or its entry into the programmed zone (no entry zone).



S= 1 No entry zone

S = 2 No exit zone

The CNC will take the dimensions of the tool into account at all times (tool offset table) to avoid it exceeding the programmed limits.

The presetting of work zones is done via Function **G22**, the programming format being:

G22 K S

In which :

*** K** Indicates the work zone you wish to define (1, 2, 3 or 4)

*** S** Indicates the enabling/disabling of the work zone:

- S=0 disabled.
- S=1 enabled as a no-entry zone.
- S=2 enabled as a no-exit zone.

On power-up, the CNC will disable all work zones. However, upper and lower limits for these zones will not undergo any variation, and they can be re-enabled through the G22 function.

4. *REFERENCE SYSTEMS*

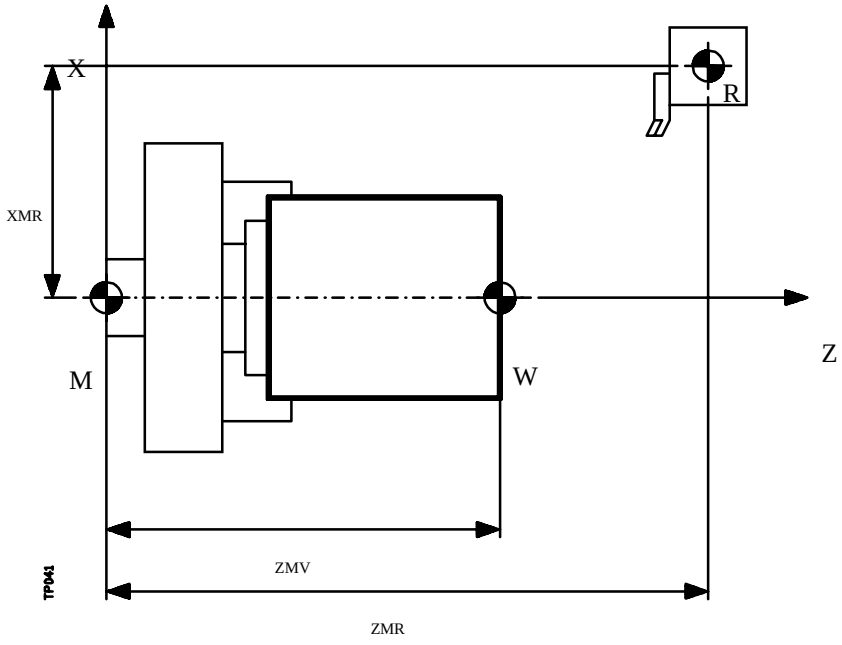
4.1 REFERENCE POINTS

A CNC machine needs the following origins and reference points defined :

* **Machine Reference Zero** or home. This is set by the manufacturer as the origin of the machine's coordinate system.

* **Part zero** or point of origin of the part. This is the point of origin which is set for programming the measurements of the part. it can be freely selected by the programmer, and its value with respect to machine zero can be set by the zero offset.

* **Machine Reference point.** This is a point on the machine established by the manufacturer around which the synchronization of the system is done. The control positions the axis on this point, instead of moving it as far as the Machine Reference Zero, taking, at this point, the reference coordinates which are defined via the axis machine parameter "REFVALUE".



M	Machine reference zero
W	Part zero
R	Machine reference point
XMW,YMW,ZMW, etc.	Coordinates of part zero
ZMR,YMR,ZMR, etc.	Coordinates of machine reference point ("REFVALUE")

4.2 MACHINE REFERENCE SEARCH (G74)

The CNC allows you to program the machine reference search in two ways:

* **MACHINE REFERENCE SEARCH OF ONE OR MORE AXES IN A PARTICULAR ORDER**

  G74 is programmed followed by the axes in which you want to carry out the reference search. For example : G74 X Z C Y

  The CNC begins the movement of all the selected axes which have a machine reference switch (machine axis parameter "DECINPUT") and in the direction indicated by the axis machine parameter "REFDIREC".

  This movement is carried out at the feedrate indicated by the axis machine parameter "REFEED1" for each axis until the home switch is hit.

  Next, the home search (marker pulse or home) will be carried out in the programmed order.

  This second movement will be carried out one axis at a time, at the feedrate indicated in the axis machine parameter "REFEED2" until the machine reference point is reached (i.e. the marker pulse is found).

* **MACHINE REFERENCE SEARCH USING THE ASSOCIATED SUBROUTINE.**

  The G74 function will be programmed alone in the block, and the CNC will automatically execute the subroutine whose number appears in the general machine parameter "REFPSUB". In this subroutine it is possible to program the machine reference searches required, and also in the required order.

In a block in which G74 has been programmed, no other preparatory function may appear.

If the machine reference search is done in JOG mode, the part zero selected is lost. The coordinates of the reference point indicated in the machine axis parameter "REFVALUE" is displayed. In all other cases, the part zero selected is maintained, so the displayed coordinates refer to this part zero.

If the G74 command is executed in MDI, the display of coordinates depends on the mode in which it is executed : Jog, Execution, or Simulation.

4.3 PROGRAMMING WITH RESPECT TO MACHINE ZERO (G53)

Function G53 can be added to any block which has path control functions.

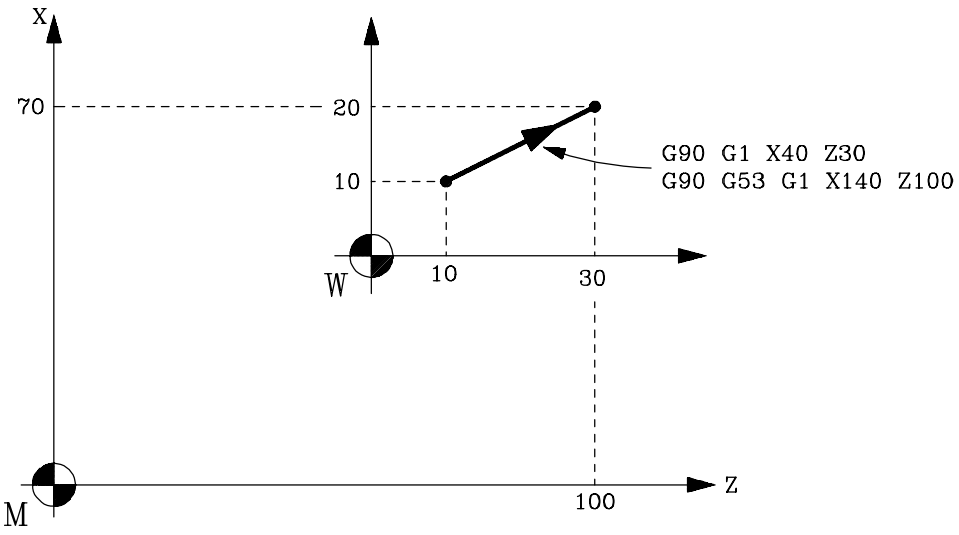
It is only used when the programming of block coordinates relating to machine zero is required. These coordinates should be expressed in millimeters or inches, depending on how the general machine parameter “INCHES” is defined.

When programming function G53 without movement information, it cancels the currently active zero offset regardless of whether it was programmed via G54-G59 or by preset (G92). The description of the zero offset preset follows:

Function G53 is not modal, so it should be programmed every time you wish to indicate the coordinates referred to machine zero.

This function temporarily cancels radius and tool length compensation.

Example when the X axis is programmed in radius:

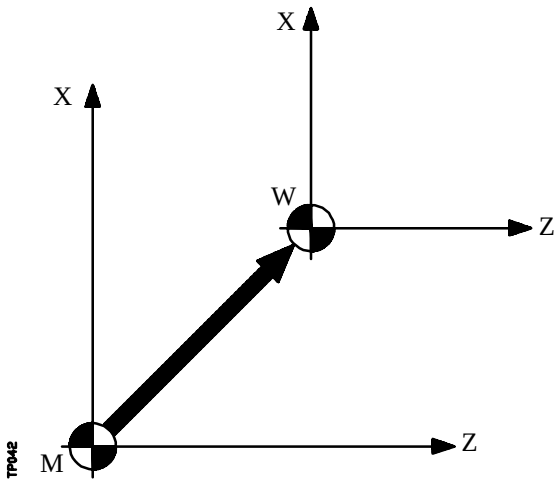


M Machine Reference (home)
W Part Zero

4.4 PRESETTING OF COORDINATES AND ZERO OFFSETS

The CNC allows you to carry out zero offsets with the aim of using coordinates related to the plane of the part, without having to modify the coordinates of the different points of the part at the time of programming.

The zero offset is defined as the distance between the part zero (point of origin of the part) and the machine zero (point of origin of the machine).



M Machine zero
W Part zero

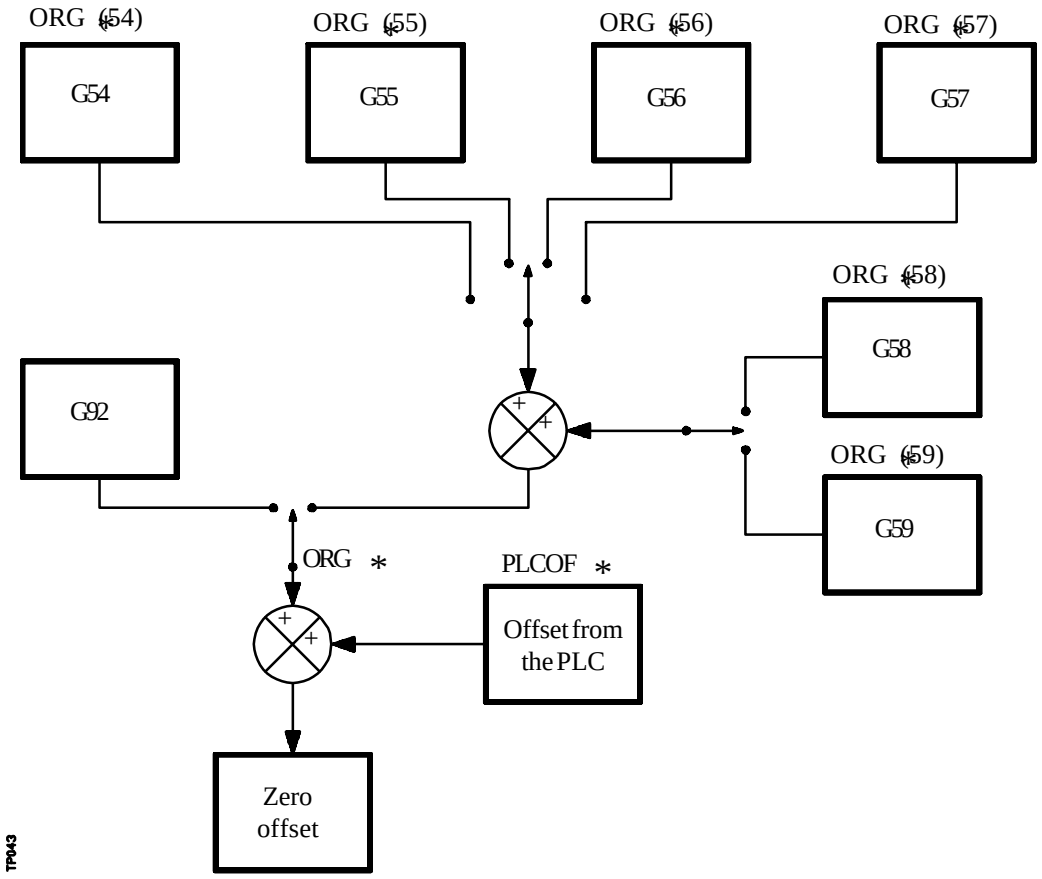
This zero offset can be carried out in one of two ways :

* Via Function G92 (coordinate preset). The CNC accepts the coordinates of the programmed axes after G92 as new axis values.

* Via the use of zero offsets (G54,G55,G56,G57,G58, G59). The CNC accepts as a new part zero the point located relative to machine zero at the distance indicated by the selected table(s).

Both functions are modal and incompatible, so if one is selected the other is disabled.

There is, moreover, another zero offset which is governed by the PLC. This offset is always added to the zero offset selected and is used (among other things) to correct deviations produced as a result of expansion, etc.


ORG (G54)
G54
ORG (G55)
G55
ORG (G56)
G56
ORG (G57)
G57
ORG (G58)
G58
G92
ORG (G59)
G59
ORG *
PLCOF *
Offset from the PLC
Zero offset
TP043

4.4.1 COORDINATE PRESET AND LIMITATION OF THE S VALUE (G92)

Via Function G92 one can select any value in the axes of the CNC, in addition to limiting the spindle speed.

* COORDINATE PRESET

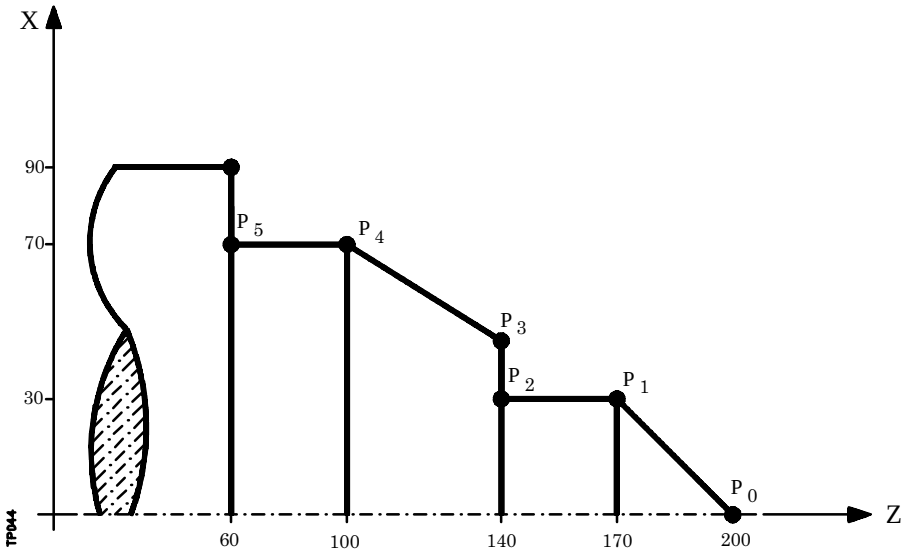
When carrying out a zero offset via Function G92, the CNC assumes the coordinates of the axes programmed after G92 as new axis values.

No other function can be programmed in the block where G92 is defined, the programming format being :

G92X...C +/- 5.5

The position values (coordinates) will be programmed in radius or diameter depending on the settings of axis machine parameter "DFORMAT".

Example:




```
G90   X50   Z200  ;  Positioning in P0
G92   X0    Z0    ;  Preset P0 as part zero
G91   X30   Z-30  ;  P1
            Z-30  ;  P2
      X20         ;  P3
      X20   Z-40  ;  P4
            Z-30  ;  P5
```

* LIMITATION OF SPINDLE SPEED

The spindle speed is limited to the value set by S5.4 by programming G92 S5.4.

This means that the CNC will not accept, from this block onwards, the programming of S values higher than the defined maximum.

Neither is it possible to exceed this maximum value from the keyboard on the front panel.

4.4.2 ZERO OFFSETS (G54..G59)

The CNC has a table of zero offsets, in which several zero offsets can be selected. The aim is to generate certain part zeros independently of the part zero active at the time.

Access to the table can be obtained from the front panel of the CNC (as explained in the Operating Manual), or via the program using high-level language commands.

These values are given in radius.

There are two kinds of zero offsets :

Absolute zero offsets (G54,G55,G56 & G57), which must be referred to machine zero.

Additive zero offsets (G58,G59).

Functions G54, G55, G56, G57, G58 & G59 must be programmed alone in the block, and work in the following way:

When one of the G54, G55, G56, G57 functions is executed, the CNC applies the zero offset programmed with respect to machine zero, cancelling the possible active zero offsets.

If one of the additive offsets G58 or G59 is executed, the CNC adds its values to the absolute zero offset active at the time. Previously cancelling the additive offset which might be active.

You can see (in the following example) the zero offsets which are applied when the program is executed.

G54 Applies zero offsets G54 ———————>G54
G58 Adds zero offsets G58 ———————————> G54+G58
G54 Cancels G58 and adds G59 ——————> G54+G59
G55 Cancels whatever and applies G55 -> G55

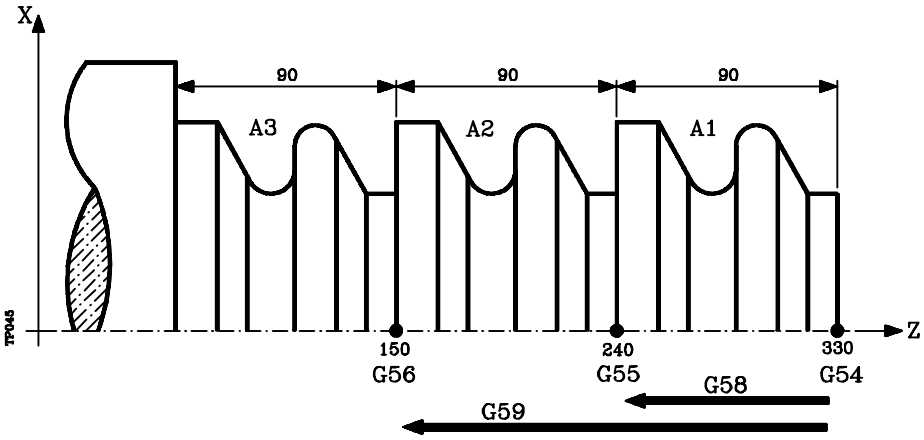
Once a Zero Offset has been selected, it will remain active until another one is selected or until a home search is carried out (G74). This Zero Offset will remain active even after powering the CNC off.

This kind of zero offsets established by program is very useful for repeated machining operations at different machine positions.

Example :

Let us assume a zero offset table initialized with the following values:

G54: X0 Z330
G55: X0 Z240 G58: Z-90
G56: X0 Z150 G59: Z-180



Using absolute coordinates.

```
G54                   ; Applies offset G54
Execution of profile  ; Executes profile A1
G55                   ; Applies offset G55
Execution of profile  ; Executes profile A2
G56                   ; Applies  G56
Execution of profile  ; Executes profile A3
```

Using incremental coordinates.

```
G54                   ; Applies offset G54
Execution of profile  ; Executes profile A1
G58                   ; Applies offset G54 + G58
Execution of profile  ; Executes profile A2
G59                   ; Applies  G54 + G59
Execution of profile  ; Executes profile A3
```

4.5 POLAR ORIGIN PRESET (G93)

Function G93 allows you to preset any point from the work plane as a new origin of polar coordinates.

This function must be programmed alone in the block, its format being :

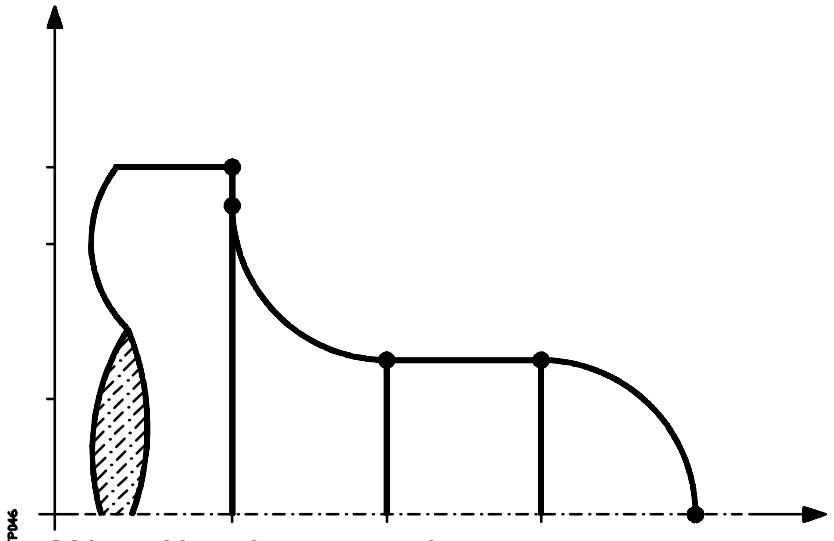
G93 I+/-5.5 J+/-5.5

Parameters I & J respectively define the abscissa and ordinate axes, of the new origin of polar coordinates.

Their values will be in radius or diameter depending on the setting of axis machine parameter “DFORMAT”

Example with X axis programming in diameter:

Assuming that the tool is at X0 Y0




```
G90  X180  Z50    ;  Point P0
G01  X160          ;  Point P1, in a straight line (G01)
G93  I90   J160    ;  Preset P5 as polar origin
G03        Q270    ;  Point P2 in an arc (G03)
G01        Z130    ;  Point P3 in a straight line (G01)
G93  I130  J0      ;  Preset P6 as new polar origin
G02        Q0      ;  Point P4 in an arc (G02)
```

If G93 is only programmed in a block, the point where the machine is at that moment becomes the polar origin.

Notes: The CNC does not modify the polar origin when defining a new part zero; but it modifies the values of the variables: "PORGF" y "PORGS".

If, while selecting the general machine parameter “PORGMOVE” a circular interpolation is programmed (G02 or G03), the CNC assumes the center of the arc as the new polar origin.

On power-up; or after executing M02, M30; or after an EMERGENCY or RESET; the CNC assumes the currently active part zero as polar origin.

When selecting a new work plane (G16, G17, G18, G19), the CNC assumes as polar origin the part zero of that plane.

5. *PROGRAMMING BY ISO CODE*

A programmed block in ISO language can consist of :

Preparatory functions **(G)**
Axis coordinates **(X...C)**
Feedrate **(F)**
Spindle speed **(S)**
Tool number **(T)**
Tool offset number **(D)**
Auxiliary functions **(M)**

This order should be maintained within each block, although it is not necessary for every block to contain the information.

The CNC allows you to program figures from 0.00001 to 99999.9999 with or without sign, working in millimeters (G71), called format #5.4, or either from 0.00001 to 3937.00787 with or without sign if the programming is done in inches (G70), called format #4.5.

Nevertheless, and in order to simplify explanations, we can say that the CNC admits Format #5.5, meaning that it admits #5.4 in millimeters and #4.5 in inches.

Any function with parameters can also be programmed in a block, apart from the number of the label or block. Thus, when the block is executed the CNC substitutes the arithmetic parameter for its value at that time.

5.1 PREPARATORY FUNCTIONS

Preparatory functions are programmed using the letter G followed by 2 digits.

They are always programmed at the beginning of the body of the block and are useful in determining the geometry and working condition of the CNC.

Table of G functions used in the CNC :

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid travel	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise circular interpolation	6.3
G03	*		*	Counter-clockwise circular interpolation	6.3
G04				Dwell/block preparation stop	7.1, 7.2
G05	*	?	*	Round corner	7.3.2
G06			*	Absolute arc center coordinates	6.4
G07	*	?		Square corner	7.3.1
G08			*	Arc tangent to previous path	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G15	*		*	C axis	6.13
G16	*		*	Selection of main plane in two directions	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.7.1
G21				Definition of upper work zone limits	3.7.1
G22			*	Activate/cancel work zones	3.7.2
G28	*		*	Selection of the second spindle or axis toggle	5.4, 7.8
G29	*	*		Selection of the main spindle or axis toggle	5.4, 7.8
G30	*		*	Spindle syncronization in position	5.5
G32	*		*	Feedrate "F" as an inverted function of time	6.14
G33	*		*	Threadcutting	6.11
G36			*	Automatic radius blend	6.9
G37			*	Tangential entry	6.7
G38			*	Tangential exit	6.8
G39			*	Automatic chamfer blend	6.10
G40	*	*		Cancellation of tool radius compensation	8.1
G41	*		*	Right-hand tool radius compensation	8.1
G42	*		*	Left-hand tool radius compensation	8.1
G45	*		*	Tangential control	6.16
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look ahead	7.4
G52			*	Movement to hard stop	6.14
G53			*	Program coordinates with respect to home	4.3

Function	M	D	V	Meaning	Section
G54	*		*	Absolute zero offset	4.4.2
G55	*		*	Absolute zero offset	4.4.2
G56	*		*	Absolute zero offset	4.4.2
G57	*		*	Absolute zero offset	4.4.2
G58	*		*	Additive zero offset	4.4.2
G59	*		*	Additive zero offset	4.4.2
G60			*	Axial drilling / tapping canned cycle	9.13
G61			*	Radial drilling / tapping canned cycle	9.14
G62			*	Longitudinal slot milling canned cycle	9.15
G63			*	Radial slot milling canned cycle	9.16
G66			*	Pattern repeat canned cycle	9.1
G68			*	Stock removal canned cycle along X axis	9.2
G69			*	Stock removal canned cycle along Z axis	9.3
G70	*	?	*	Programming in inches	3.3
G71	*	?		programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G74			*	Machine reference search	4.2
G75			*	Probing until touching	10.1
G76			*	Probing while touching	10.1
G77	*		*	Slaved axis	7.7.1
G77S	*		*	Spindle synchronization in speed	5.5
G78	*	*		Slaved axis cancellation	7.7.2
G78S	*	*		Cancel spindle synchronization	5.5
G81			*	Turning canned cycle with straight sections	9.4
G82			*	Facing canned cycle with straight sections	9.5
G83			*	Drilling canned cycle	9.6
G84			*	Turning canned cycle with circular sections	9.7
G85			*	Facing canned cycle with circular sections	9.8
G86			*	Longitudinal threadcutting canned cycle	9.9
G87			*	Face threadcutting canned cycle	9.10
G88			*	Grooving canned cycle along X axis	9.11
G89			*	Grooving canned cycle along Z axis	9.12
G90	*	?		Programming in absolute	3.4
G91	*	?	*	Programming in incremental	3.4
G92				Coordinate preset/spindle speed limit	4.4.1
G93				Polar origin	4.5
G94	*	?		Feedrate in millimeters(inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters(inches) per revolution	5.2.2
G96	*		*	Constant Surface Speed	5.3.1
G97	*	*		Spindle speed in r.p.m.	5.3.2

M means MODAL, i.e., that once programmed, the G function remains active as long as another incompatible G function is not programmed, M02, M30, EMERGENCY, RESET are not programmed or the CNC is not turned on or off.

Letter **D** means BY DEFAULT, i.e., that these will be assumed by the CNC when turned on, after executing M02, M30 or after EMERGENCY or RESET.

In cases indicated with **?** it must be interpreted that the BY DEFAULT of these G functions depends on the settings of the general CNC machine parameters.

V means that the G function is displayed next to the machining conditions in the execution and simulation modes.

5.2 FEEDRATE F

The machining feedrate can be selected from the program. It remains active until another feedrate is programmed.

It is represented by the letter **F**. Depending on whether it is working in G94 or G95, it is programmed in mm/minute (inches/minute) or in mm/revolution (inches/revolution).

It's programming format is 5.5 in mm. and 4.5 in inches.

The maximum operating feedrate of the machine, limited on each axis by the axis machine parameter "MAXFEED", may be programmed via code F0, or by giving F the corresponding value.

The programmed feedrate F is effective working in linear (G01) or circular (G02, G03) interpolation. If function F is not programmed, the CNC assumes the feedrate to be F0. When working in rapid travel (G00), the machine will move at the rapid feedrate indicated by the axis machine parameter "G00FEED", apart from the F programmed.

The programmed feedrate F may be varied between 0% and 255% via the PLC, or by DNC, or between 0% and 120% via the switch located on the Operator Panel of the CNC.

The CNC, however, is equipped with the general machine parameter "MAXFOVR" to limit maximum feedrate variation.

If you are working in rapid travel (G00), rapid feedrate will be fixed at 100%, alternatively it can be varied between 0% and 100%, depending on how the machine parameter "RAPIDOVR" is set.

When functions G33 (electronic threading), G86 (longitudinal threadcutting canned cycle) or G87 (face threadcutting canned cycle) are executed, the feedrate cannot be modified. It is set at 100% of programmed F.

5.2.1 FEEDRATE IN MM/MIN OR INCHES/MIN (G94)

From the moment the code G94 is programmed, the control takes that the feedrates programmed through **F5.5** are in mm/min or inches/mm.

If the movement corresponds to a rotary axis, the CNC interprets the feedrate as being programmed in degrees/min.

If an interpolation is made between a rotary and a linear axis, the programmed feedrate is taken in mm/min or inches/min, and the movement of the rotary axis (programmed in degrees) will be considered programmed in millimeters or inches.

The relationship between the feedrate of the axis component and the programmed feedrate "F" is the same as that between the movement of the axis and the resulting programmed movement.

$$\text{Feedrate component} = \frac{\text{Feedrate F} \times \text{Movement of axis}}{\text{Resulting programmed movement}}$$

Example :

On a machine which has linear X and Z axes and rotary C axis, all located at point X0 Y0 C0, the following movement is programmed :

G1 G90 X100 Z20 C270 F10000

You get:

$$Fx = \frac{F\ \triangle x}{\sqrt{(\triangle x)^2+(\triangle y)^2+(\triangle c)^2}} \quad \frac{10000 \times 100}{\sqrt{100^2 + 20^2 + 270^2}}$$

$$Fz = \frac{F\ \triangle y}{\sqrt{(\triangle x)^2+(\triangle y)^2+(\triangle c)^2}} \quad \frac{10000 \times 20}{\sqrt{100^2 + 20^2 + 270^2}}$$

$$Fc = \frac{F\ \triangle c}{\sqrt{(\triangle x)^2+(\triangle y)^2+(\triangle c)^2}} \quad \frac{10000 \times 270}{\sqrt{100^2 + 20^2 + 270^2}}$$

Function G94 is modal i.e. once programmed it stays active until G95 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to how the general machine parameter “IFEED” is set.

5.2.2 FEEDRATE IN MM/REV.OR INCHES/REV (G95)

From the moment when the code G95 is programmed, the control assumes that the feedrates programmed through **F5.5** are in mm/rev or inches/mm.

This function does not affect the rapid moves (G00) which will be made in mm/min or inch/min. By the same token, it will not be applied to moves made in the JOG mode, during tool inspection, etc.

Function G95 is modal i.e. once programmed it stays active until G94 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to the general machine parameter “IFEED”.

5.3 SPINDLE SPEED AND SPINDLE ORIENTATION (S)

The turning speed of the spindle is programmed directly in rpm via code S5.4 if in G97 or in m/min. (feet/min when working in inches) if in G96.

The maximum value is limited by spindle machine parameters “MAXGEAR1”, MAXGEAR2, MAXGEAR 3 and MAXGEAR4", in each case depending on the spindle range selected.

It is also possible to limit this maximum value from the program by using function G92 S5.4.

The programmed turning speed S may be varied from the PLC, DNC, or by the SPINDLE keys “+” and “-” on the Operator Panel of the CNC.

This speed variation is made between the maximum and minimum values established by spindle machine parameters “MINSOVR and MAXSOVR”.

The incremental pitch associated with the SPINDLE keys “+” and “-” on the CNC Operator Panel in order to vary the programmed S value is fixed by the spindle machine parameter “SOVRSTEP”.

When functions G33 (electronic threading), G86 (longitudinal threadcutting canned cycle) or G87 (face threadcutting canned cycle) are executed, the spindle speed cannot be modified. It is set at 100% of programmed S.

5.3.1 CONSTANT SURFACE SPEED (G96)

When G96 is programmed, the CNC understands that the spindle speed (programmed by means of **S5.4**) is given in m/min. or feet/min. and the lathe begins to work in Constant Surface Speed (C.S.S.) mode.

It is recommended to program the spindle speed S5.4 in the same block with the G96 being the spindle speed range (M41, M42, M43, M44) selected either in the same block or in the previous one.

If a spindle speed is not programmed and a there is none selected previously, or the corresponding spindle speed range (gear) has not been selected, the CNC will issue the corresponding error message.

Function G96 is modal, that is that once programmed, it will remain active until a G97 is programmed.

On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC assumes function G97.

5.3.2 SPINDLE ROTATION SPEED IN RPM (G97)

When G97 is programmed, the CNC assumes that the spindle speed programmed by **S5.4** is given in revolutions per minute.

If in the block containing G97 the spindle speed S5.4 is not programmed, the CNC assumes the current spindle speed as the one programmed.

Function G97 is modal i.e. once programmed it is active until G96 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G97.

5.4 SPINDLE SELECTION (G28, G29)

The lathe model may have 2 spindle: the main and the second one.
Both spindle may be turning at the same time; but the CNC will only control one at a time.

This selection is made by means of functions G28 and G29.

G28: Selects the second spindle
G29: Selects the main spindle

Once the desired spindle has been selected, it may be manipulated from the CNC keyboard or by the following functions:

M3, M4, M5, M19
S****
G33, G94, G95, G96, G97

Both spindle can work either in open or closed loop; but only the main spindle can work as a "C" axis

Functions G28 and G29 are modal and incompatible with each other.

G28 and G29 must be programmed alone in the block. That is the only information this block may contain.

On power-up, after executing an M02, M30 or after an EMERGENCY or RESET, the CNC assumes function G29 (main spindle selected).

Dual spindle operating example.

On power-up, the CNC always selects the main spindle.
All the keyboard actions and by spindle related functions affect the main spindle.
Example: S1000 M3 Main spindle clockwise at 1000 rpm

To select the second spindle, execute function G28.
From then on, All the keyboard actions and spindle related functions affect the second spindle.
The main spindle remains in its previous status.
Example: S1500 M4 Second spindle turns counter-clockwise at 1500 rpm.
The main spindle keeps turning at 1000 rpm

To select the main spindle again, execute function G29.
From then on, all the keyboard actions and spindle related functions affect the main spindle.
The second spindle stays in its previous status.
Example: S2000 The main spindle keeps turning clockwise but at 2000 rpm.
The second spindle keeps turning at 1500 rpm.

5.5 SYNCHRONIZED SPINDLES (G30, G77S, G78S)

With function G77S, two spindles (main and secondary) may be synchronized in speed. This synchronism may be cancelled with function G78S.

Always program G77S and G78S because functions G77, G78 to slave and unslave the axes.

When the spindles are synchronized in speed, the second one turns at the same speed as the main spindle.

Function G77S may be executed at any time, open loop (M3, M4) or closed loop (M19), the spindles may even have different ranges (gears).

General output "SYNSPEED (M5560)" will be high while the spindle are in synch (same speed).

When this synchronism is cancelled (G78S), the second spindle recovers its previous speed and status (M3, M4, M5, M19) and the main spindle stays in the current status.

If while in synchronism, an S is programmed greater than the maximum allowed, the CNC applies the maximum value while they are synchronized. When cancelling this synchronism, the limit is no longer applied and the main spindle assumes the programmed speed.

While the spindles are synchronized in speed, function G77S active, with G30 they may also be synchronized in position and set an angular offset between them so the second spindle follows the main spindle at this set offset distance.

Programming format: G30 D ±359.9999 (offset in degrees)

For example, with G30 D90 the second spindle will turn 90º behind the main spindle.

Considerations:

Before activating the synchronism, both spindles must be homed (referenced).
The spindles cannot be synchronized if the XC or ZC (C axis) plane is selected.
To synchronized the spindles in position (G30) they must be synchronized in speed already (G77S).

While the spindles are synchronized, only the signals of the main spindle will be attended to PLCCNTL, SPDLINH, SPDLREV, etc. On the other hand, when making a thread, only the feedback and reference pulses of the main spindle will be taken into account.

While the spindle synchronism is active, it is possible to:
- Execute functions G94, G95, G96, G97, M3, M4, M5, M19 S***
- Change the spindle speed via DNC, PLC or CNC (S)
- Change the spindle speed override via DNC, PLC, CNC or keyboard
- Change the spindle speed limit via DNC, PLC or CNC (G92 S)

But, it is **NOT** possible to:
- Toggle the spindles: G28, G29
- Change gears: M41, M42, M43, M44.
- Define the XC or ZC plane in "C" axis mode.

5.6 TOOL NUMBER (T) AND TOOL OFFSET (D)

With the "T" function, it is possible to select the tool and with the "D" function it is possible to select the offset associated with it. When defining both parameters, the programming order is T D. For example: T6 D17

If the machine has a tool magazine, the CNC looks up the "Tool magazine table" to know the position occupied by the selected tool and the desired one.

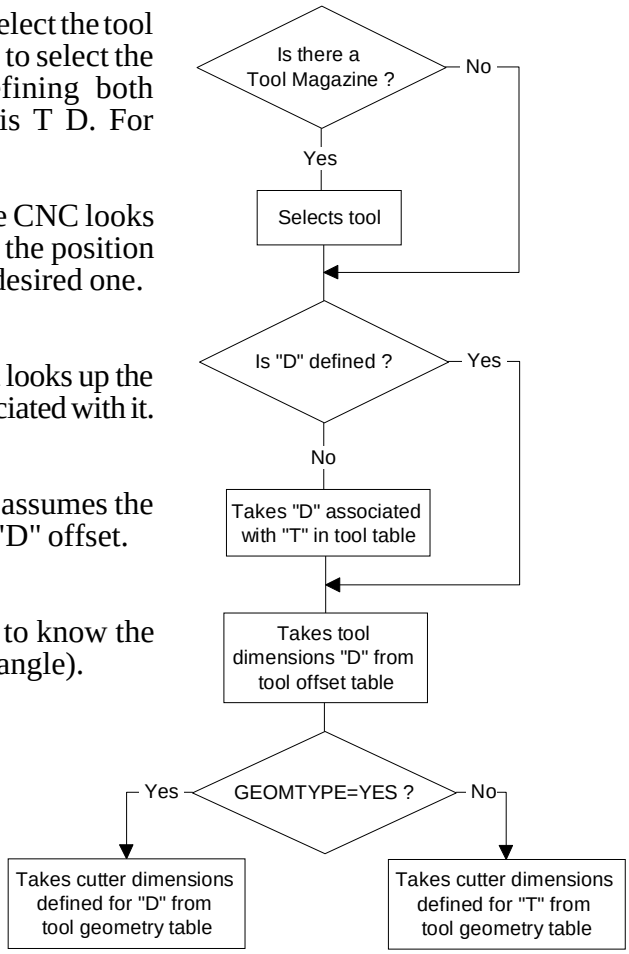
If the "D" function has not be defined, it looks up the "Tool table" to know the "D" offset associated with it.

It looks up the "tool offsets table" and assumes the tool dimensions corresponding to the "D" offset.

It looks up the "Tool geometry table" to know the cutter geometry (width, angle, cutting angle).

The "Tool geometry table is associated with the T or the D according to the machine manufacturer's criteria, general machine parameter "GEOMTYPE (P123)"

To access, check and define these tables, refer to chapter 6 of the Operating Manual.


Is there a Tool Magazine ?
No
Yes
Selects tool
Is "D" defined ?
Yes
No
Takes "D" associated with "T" in tool table
Takes tool dimensions "D" from tool offset table
GEOMTYPE=YES ?
Yes
No
Takes cutter dimensions defined for "D" from tool geometry table
Takes cutter dimensions defined for "T" from tool geometry table

How to use the T and D functions

- The "T" and "D" functions may be programmed alone or together as shown in the following example:

  T5 D18 selects tool 5 and assumes the dimensions of tool offset 18
  D22 Tool 5 stays selected and it assumes the dimensions of tool offset 22
  T3 selects tool 3 and assumes the dimensions of the offset associated with that tool.

- When having a tool turret, it is rather common to use more tools than the number of tool positions of the turret. Thus, the same turret position must be used by more than one tool.

  In those cases, both "T" and "D" must be programmed.

  The "T" function refers to the turret position and the "D" function to the dimensions of the tool located in that position.

  Thus, for example, programming T5 D23 means selecting the turret position 5 and assuming the geometry and dimensions of tool offset 23.

- When having a tool holding arm with 2 cutters, both "T" and "D" must be programmed as well.

  The "T" function refers to the arm and the "D" function to the cutter dimensions.

  Thus, one may program T1 D1 or T1 D2 depending on which of the 2 cutters is to be used.

Tool length and radius compensation.

The CNC looks up the "tool offset table" and assumes the tool dimensions corresponding to the active "D" offset.

Length compensation is applied at all times, whereas radius compensation must be selected by the operator by means of functions G40, G41, G42.

If there is no tool selected or D0 is defined, neither tool length nor radius compensation is applied.

For further information, refer to chapter 8 "tool compensation" in this manual..

5.7 MISCELLANEOUS FUNCTION (M)

The miscellaneous functions are programmed by means of the **M4** Code, it being possible to program up to 7 functions in the same block.

When more than one function has been programmed in one block, the CNC executes these correlatively to the order in which they have been programmed.

The CNC is provided with an M functions table with "NMISCFUN" (general machine parameter) components, specifying for each element:

* The number (0-9999) of the defined miscellaneous M function.
* The number of the subroutine which is required to associate to this miscellaneous function.
* An indicator which determines if the M function is executed before or after the movement block in which it is programmed.
* An indicator which determines if the execution of the M function interrupts block preparation or not.
* An indicator which determines if the M function is executed or not, after the execution of the associated subroutine.
* An indicator which determines if the CNC must wait for the signal AUX END or not (Executed M signal, coming from the PLC), to continue the execution of the program.

If, when executing the M miscellaneous function, this is not defined in the M functions table, the programmed function will be executed at the beginning of the block and the CNC will wait for the AUX END to continue the execution of the program.

Some of the miscellaneous functions are assigned an internal meaning in the CNC.

Warning:

All the miscellaneous "M" functions which have an associated subroutine must be programmed alone in a block.

If, while executing the associated subroutine of an "M" miscellaneous function, there is a block containing the same "M", this will be executed but not the associated subroutine.

5.7.1 M00. PROGRAM STOP

When the CNC reads code M00 in a block, it interrupts the program. To start up again, press CYCLE START.

We recommend that you set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.7.2 M01. CONDITIONAL PROGRAM STOP

This is identical to M00, except that the CNC only takes notice of it if the signal M01 STOP from the PLC is active (high logic level).

5.7.3 M02. END OF PROGRAM

This code indicates the end of program and carries out a “General Reset” function of the CNC (returning it to original state). It also carries out the M05 function.

It is recommended to set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.7.4 M30. END OF PROGRAM WITH RETURN TO FIRST BLOCK

Identical to M02 except that the CNC returns to the first block of the program.

5.7.5 M03. CLOCKWISE SPINDLE ROTATION

This code represents clockwise spindle start.

It is recommended to set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.7.6 M04. COUNTERCLOCKWISE SPINDLE ROTATION

This code represents counter-clockwise spindle start. We recommend that you set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.7.7 M05. SPINDLE STOP

It is recommended to set this function in the table of M functions, so that it is executed at the end of the block in which it is programmed.

5.7.8 M06. TOOL CHANGE

If the general machine parameter “TOFFM06” (indicating that it is a machining center) is active, the CNC sends instructions to the tool changer and updates the table corresponding to the tool magazine.

It is recommended to set this function in the table of M functions, so that the subroutine corresponding to the tool changer installed in the machine is executed.

5.7.9 M19. SPINDLE ORIENTATION

With this CNC it is possible to work with the spindle in open loop (M3, M4) and with the spindle in closed loop (M19).

In order to work in closed loop, it is necessary to have a rotary encoder installed on the spindle of the machine.

To switch from open loop to closed loop, execute function M19 or M19 S±5.5. The CNC will act as follows:

* If the spindle does not have a home switch, the CNC changes the spindle speed until it reaches the one set by spindle machine parameter "REFEED2; finds the marker pulse (home) and, then, orients the spindle to the position defined by S±5.5.

* If the spindle has a home switch, the CNC modifies the spindle speed until it reaches the one set by spindle machine parameter "REFEED1". Then, it carries out the search for the home switch at this speed. Next, it looks for the marker pulse (home) at the speed set by spindle machine parameter "REFEED2" and, finally, it orients the spindle to the position defined by S±5.5.

If only M19 is executed, the spindle is oriented to position "S0" after having "found" the home switch.

To, now, orient the spindle to another position, program M19 S±5.5, the CNC will not perform the home search since it is already in closed loop and it will orient the spindle to the indicated position. (S±5.5).

The S±5.5 code indicates the spindle orient position, in degrees, from the encoder's marker pulse position (S0).

The sign indicates the counting direction and the 5.5 value is always considered to be absolute coordinates regardless of the type of units currently selected.

Example:

S1000 M3	Spindle in open loop
M19 S100	The spindle switches to closed loop. Home search and positioning (orientation) at 100º
M19 S-30	The spindle orients to -30º, passing through 0º.
M19 S400	The spindle turns a whole revolution and positions at 40º.

5.7.10 M41, M42, M43, M44. SPINDLE SPEED RANGE CHANGE

The CNC offers 4 spindle speed ranges M41, M42, M43 and M44 with maximum speed limits set by the spindle machine parameters “MAXGEAR1”, MAXGEAR2", “MAXGEAR3” and “MAXGEAR4”.

If machine parameter “AUTOGEAR” is set so the CNC executes the range change automatically, M41 thru M44 will be sent out automatically by the CNC without having to be programmed.

If this machine parameter is set for non-automatic gear change, M41 thru M44 will have to be programmed every time a gear change is required. Bear in mind that the maximum voltage value assigned to machine parameter “MAXVOLT” corresponds to the maximum speed indicated for each one of the speed ranges (machine parameters “MAXGEAR1” thru “MAXGEAR4”).

5.7.11 M45 AUXILIARY SPINDLE / LIVE TOOL

In order to use this miscellaneous function, it is necessary to set one of the axes of the machine as auxiliary spindle or live tool (general machine parameter P0 thru P7).

To use the auxiliary spindle or live tool, execute the command: **M45 S±5.5** where **S** indicates the turning speed in rpms and the sign indicates the turning direction.

The CNC will output the analog voltage corresponding to the selected speed according to the value assigned to the machine parameter "**MAXSPEED**" for the auxiliary spindle.

To stop the auxiliary spindle, program **M45** or **M45 S0**.

Whenever the auxiliary spindle or live tool is active, the CNC will let the PLC know by activating the general logic output "**DM45**" (M5548).

Also, it is possible to set the machine parameter for the auxiliary spindle "**SPDLOVR**" so the Override keys of the front panel can modify the currently active turning speed of the auxiliary spindle.

6. *PATH CONTROL*

The CNC allows you to program movements on one axis only or several at the same time.

Only those axes which intervene in the required movement are programmed. The programming order of the axes is as follows :

X, Y, Z, U, V, W, A, B, C

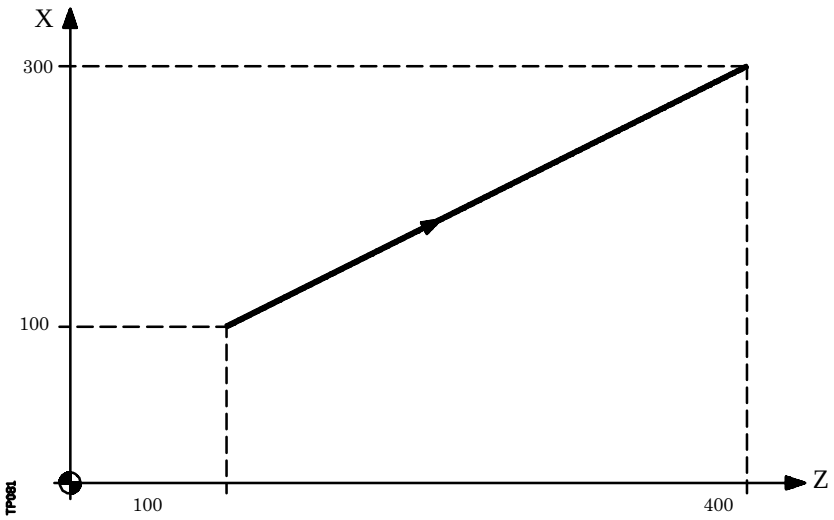
The coordinates for each axis are programmed in radius or diameter depending on the setting of their corresponding axis-machine parameter “DFORMAT”

6.1 RAPID TRAVEL (G00)

The movements programmed after G00 are executed at the rapid feedrate indicated in the axis machine parameter "G00FEED".

Independently of the number of axis which move, the resulting path is always a straight line between the starting point and the final point.

Example with X axis programmed in radius:




```
                X100  Z100  ;  Starting point
G00  G90  X300  Z400  ;  Programmed path
```

It is possible, via the general machine parameter "RAPIDOVR", to establish if the feedrate override % switch (when working in G00) operates from 0% to 100%, or whether it stays constant at 100%.

When G00 is programmed, the last "F" programmed is not cancelled i.e. when G01, G02 or G03 are programmed again "F" is recovered.

G00 is modal and incompatible with G01, G02, G03, G33 and G75. Function G00 can be programmed as G or G0.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

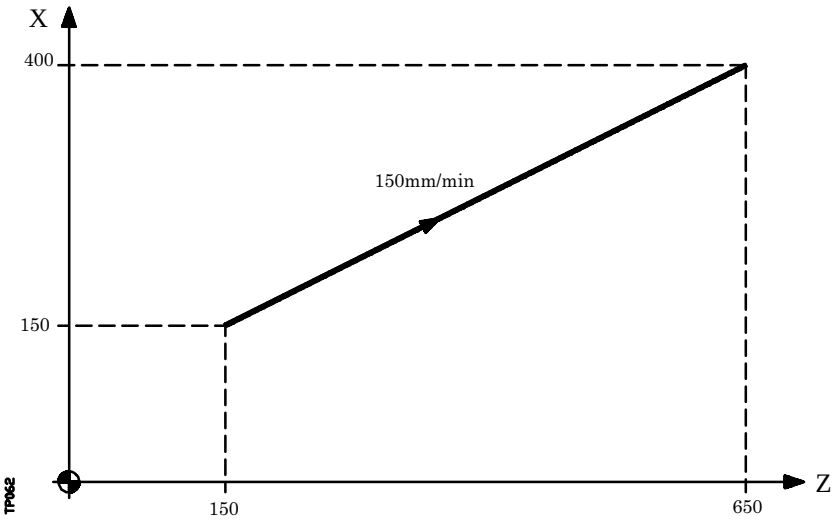
6.2 LINEAR INTERPOLATION (G01)

The movements programmed after G01 are executed according to a straight line and at the programmed feedrate "F".

When two or three axes move simultaneously the resulting path is a straight line between the starting point and the final point.

The machine moves according to this path to the programmed feedrate "F". The CNC calculates the feedrates of each axis so that the resulting path is the "F" value programmed.

Example with X axis programmed in diameter:



G01 G90 X800 Z650 F150

The programmed feedrate "F" may vary between 0% and 120% via the switch located on the Control Panel of the CNC, or by selecting between 0% and 255% from the PLC, or via the DNC or the program.

Nevertheless, the CNC has general machine parameter "MAXFOVR" to limit maximum variation of the feedrate.

It is possible to program several positioning-only axes in a linear interpolation block. The CNC will calculate the feedrate for these positioning-only axes so they reach the target coordinate at the same time as the interpolating axes.

Function G01 is modal and incompatible with G00, G02, G03, G33 and G75. Function G01 can be programmed as G1.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.3 CIRCULAR INTERPOLATION (G02. G03)

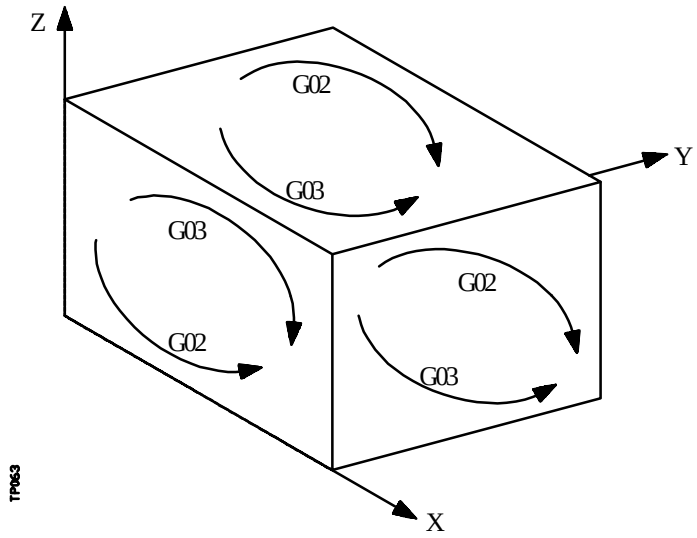
There are two ways of carrying out circular interpolation:

G02: Clockwise circular interpolation

G03: Counter-clockwise circular interpolation

Movements programmed after G02 and G03 are executed in the form of a circular path and at the programmed feedrate “F”.

Clockwise (G02) and counter-clockwise (G03) definitions are established according to the system of coordinates shown below:



This system of coordinates refers to the movement of the tool on the part.

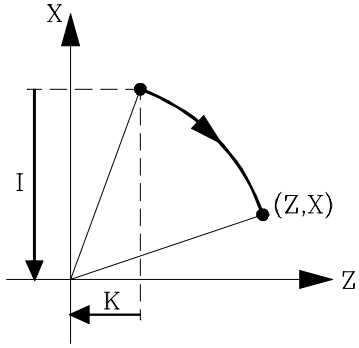
Circular interpolation can only be executed on a plane. The form of definition of circular interpolation is as follows :

a) CARTESIAN COORDINATES

The coordinates of the endpoint of the arc and the position of the center with respect to the starting point are defined according to the axes of the work plane.

The center coordinates (which should always be programmed even if they have "0" value) are defined in radius by means of the letters I, J, or K, each one of these being associated to the axes as follows:

Axes X,U,A —> I
Axes Y,V,B —> J
Axes Z,W,C —> K



Programming format:

Plane XY: G02(G03) X±5.5 Y±5.5 I±5.5 J±5.5
Plane ZX: G02(G03) X±5.5 Z±5.5 I±5.5 K±5.5
Plane YZ: G02(G03) Y±5.5 Z±5.5 J±5.5 K±5.5

The programming order of the axes is always maintained regardless of the plane selected,, as are the respective center coordinates.

Plane AY: G02(G03) Y±5.5 A±5.5 J±5.5 I±5.5
Plane XU: G02(G03) X±5.5 U±5.5 I±5.5 I±5.5

b) POLAR COORDINATES

It is necessary to define the angle to be travelled **Q** and the distance from the starting point to the center (optional), according to the axes of the work plane.

The center coordinates are defined in radius by means of the letters I, J, or K, each one of these being associated to the axes as follows:

Axes X,U,A —> I
Axes Y,V,B —> J
Axes Z,W,C —> K

If the center of the arc is not defined, the CNC will assume it that this coincides with the current polar origin.

Programming format:

Plane XY: G02(G03) Q±5.5 I±5.5 J±5.5
Plane ZX: G02(G03) Q±5.5 I±5.5 K±5.5
Plane YZ: G02(G03) Q±5.5 J±5.5 K±5.5

c) CARTESIAN COORDINATES WITH RADIUS PROGRAMMING

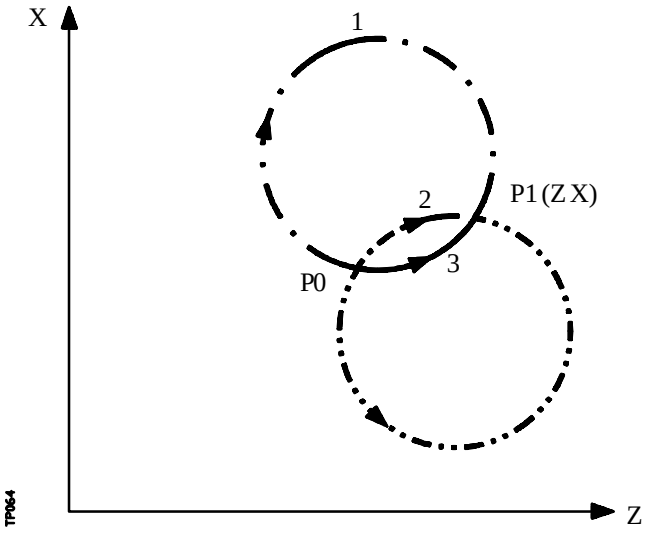
The coordinates of the endpoint of the arc and radius **R** are defined.

Programming format:

Plane XY:	G02(G03)	X±5.5	Y±5.5	R±5.5
Plane ZX:	G02(G03)	X±5.5	Z±5.5	R±5.5
Plane YZ:	G02(G03)	Y±5.5	Y±5.5	R±5.5

If a complete circle is programmed, with radius programming, the CNC will show the corresponding error, as infinite solutions exist.

If an arc is less than 180º, the radius is programmed with a plus sign, and a minus sign if it is more than 180º.

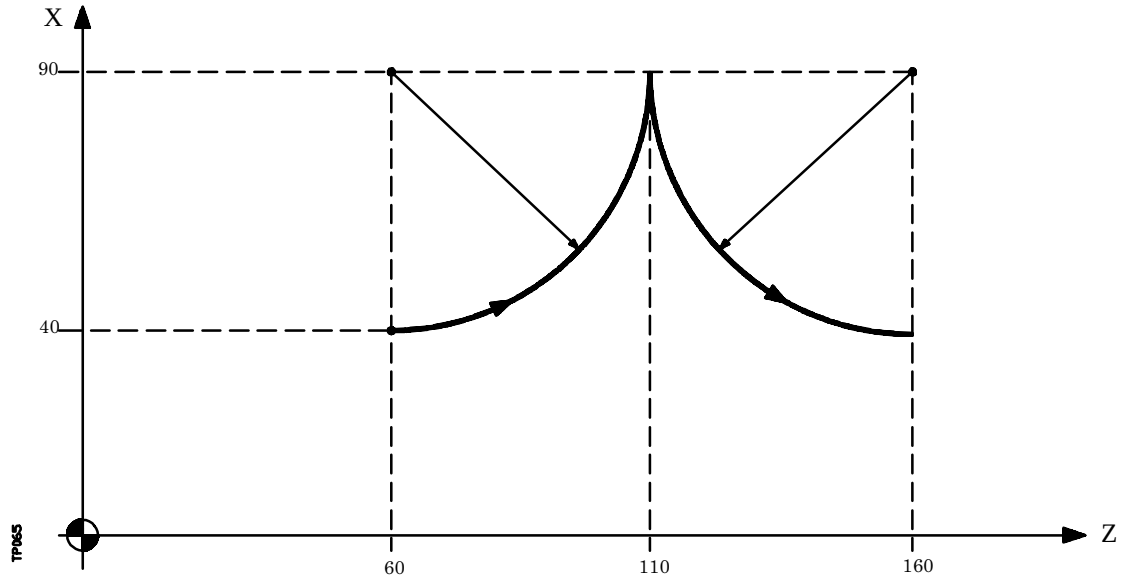


If P0 is the starting point and P1 the endpoint, there are 4 arcs which have the same value passing through both points.

Depending on the circular interpolation G02 or G03, and on the radius sign, the relevant arc is defined. Thus the programming format of the sample arcs is as follows:

Arc 1 G02 X.. Z.. R -..
Arc 2 G02 X.. Z.. R +..
Arc 3 G03 X.. Z.. R +..
Arc 4 G03 X.. Z.. R -..

Example with X axis programmed in radius:



Various programming modes are analyzed below, point X40 Z60 being the starting point.

Cartesian coordinates:

```
G90  G03  X90  Z110  I50  K0
          X40  Z160  I0   K50
```

Polar coordinates:

```
G90  G03  Q0     I50  K0
          Q-90   I0   K50
```

or:

```
G93  I90  J60   ;  defines polar center
G03  Q0
G93  I90  J160  ;  defines new polar center
Q-90
```

Cartesian coordinates with radius programming:

```
G90  G03  X90  Z110  R50
          X40  Z160  R50
```

The CNC calculates, depending on the programmed arc, the radii of the starting point and endpoint. Although in theory both points should be exactly the same, the CNC enables you to select with the general machine parameter "CIRINERR", the maximum difference permissible between both radii, If this value is exceeded, the CNC displays the corresponding error.

The programmed feedrate "F" can be varied between 0% and 120% by using the switch located on the Operator Panel of the CNC, or by selecting it between 0% and 255% from the PLC, via the DNC or from the program.

The CNC, however, has general machine parameter "MAXFOVR" to limit the maximum variation of the feedrate.

If the general machine parameter "PORGMOVE" has been selected and a circular interpolation (G02 or G03) is programmed, the CNC assumes the center of the arc to be a new polar origin.

Functions G02 and G03 are modal and incompatible both among themselves and with G00, G01, and G33. Functions G02 and G03 can be programmed as G2 and G3.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

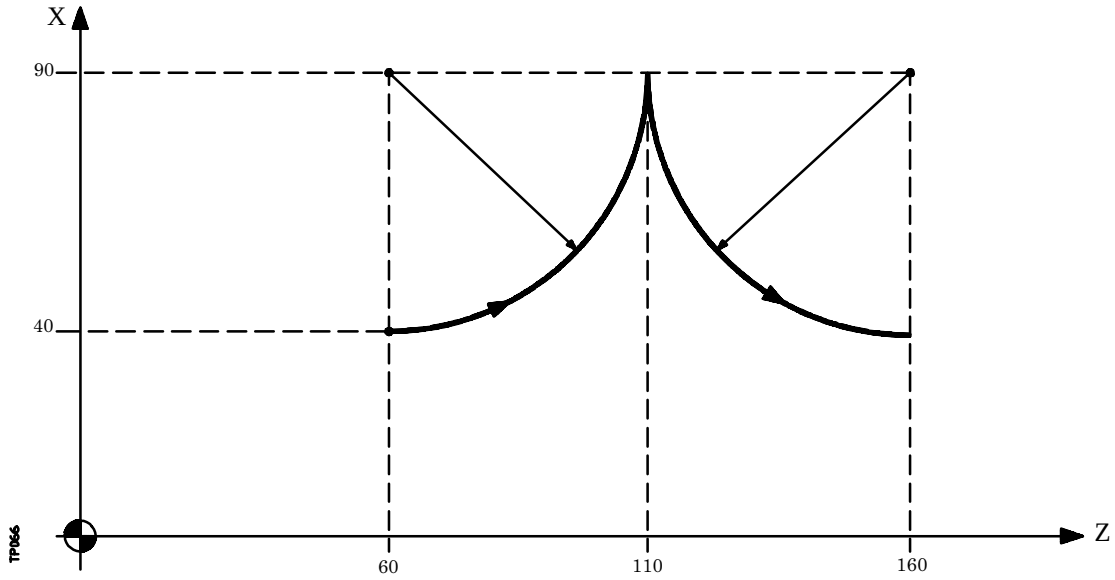
6.4 CIRCULAR INTERPOLATION BY PROGRAMMING THE CENTER OF THE ARC IN ABSOLUTE COORDINATES (G06)

By adding function **G06** to a circular interpolation block you can program the coordinates of the center of the arc (I,J, or K) in absolute coordinates i.e. with respect to the zero origin and not to the beginning of the arc.

The center coordinates will be programmed in radius or diameter depending on the programming units selected by axis-machine parameter "DFORMAT".

Function G06 is not modal, so it should be programmed any time the coordinates of the center of the arc are required in absolute coordinates. G06 can be programmed as G6.

Example with X axis programmed in radius:



Various programming modes are analyzed below, point X40 Z60 being the starting point.

Cartesian coordinates:

G90 **G06** G03 X90 Z110 **I90** **K60**
G06 X40 Z160 **I90** **K160**

Polar coordinates:

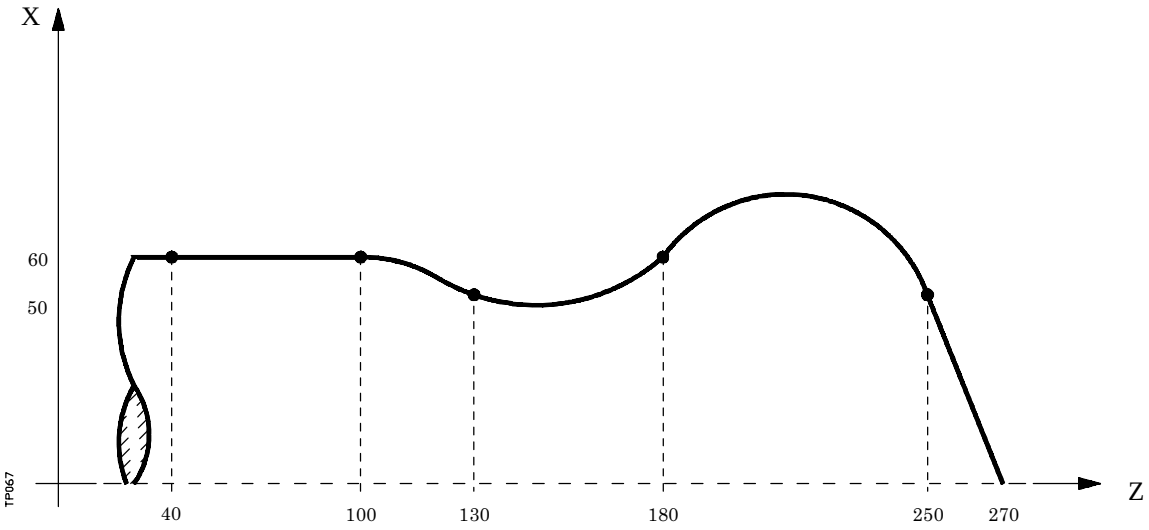
G90 **G06** G03 Q0 **I90** **KG0**
G06 Q-90 **I90** **K160**

6.5 ARC TANGENT TO THE PREVIOUS PATH (G08)

Via function **G08** you can program an arc tangential to the previous path without having to program the coordinates (I.J &K) of the center.

Only the coordinates of the endpoint of the arc are defined, either in polar coordinates or in cartesian coordinates according to the axes of the work plane.

Example with X axis programmed in radius:




```
G90  G01  X0   Z270
          X50  Z250
     G08  X60  Z180  ;  arc tangential to previous path
     G08  X50  Z130  ;  arc tangential to previous path
     G08  X60  Z100  ;  arc tangential to previous path
     G01  X60  Z40
```

Function G08 is not modal, so it should always be programmed if you wish to execute an arc tangential to the previous path. Function G08 can be programmed as G8.

Function G08 enables the previous path to be a straight line or an arc and does not alter its history. The same function G01, G02 or G03 stays active after the block is finished.

Warning:

When using function G08 it is not possible to execute a complete circle, as an infinite range of solutions exists. The CNC displays the corresponding error code.

6.6 ARC DEFINED BY THREE POINTS (G09)

Through function **G09** you can define an arc by programming the endpoint and an intermediate point (the starting point of the arc is the starting point of the movement). In other words, instead of programming the coordinates of the center, you program any intermediate point.

The endpoint of the arc is defined in cartesian or polar coordinates, and the intermediate point is always defined in Cartesian coordinates by the letters I,J, or K, each one being associated to the axes as follows:

Axes X,U,A —> I
Axes Y,V,B —> J
Axes Z,W,C —> K

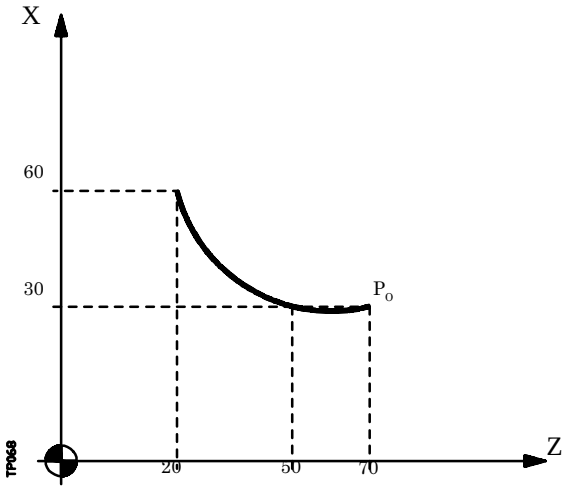
In Cartesian coordinates: G18 **G09 X±5.5 Z±5.5 I±5.5 K±5.5**

Polar coordinates: G18 **G09 R±5.5 Q±5.5 I±5.5 K±5.5**

Example being P0 the initial point:

X axis programmed in radius: **G09 X60 Z20 I30 K50**

X axis programmed in diameter: **G09 X120 Z20 I60 K50**



Function G09 is not modal, so it should always be programmed if you wish to execute an arc defined by three points. Function G09 can be programmed as G9.

When G09 is programmed it is not necessary to program the direction of movement (G02 or G03).

Function G09 does not alter the history of the program. The same G01, G02 or G03 function stays active after finishing the block.

Warning:

When using function G09 it is not possible to execute a complete circle, as you have to program three different points. The CNC displays the corresponding error code.

6.7 HELICAL INTERPOLATION

A helical interpolation consists of a circular interpolation in the work plane and the movement of the rest of the programmed axes.

It is usually utilized on special machines having an auxiliary axis.

The helical interpolation is programmed in a block.

The circular interpolation by means of functions: G02, G03, G08 or G09.
The movement of the other axis or axes by indicating the target coordinate of that movement.

Example of circular interpolation in the YZ plane and movement of the X axis:

G02 Y Z J K X

For the helical interpolation to make more than one turn, the circular interpolation and the movement of ONLY one axis must be programmed.

The helical pitch must also be defined (5.5 format) using the letters I, J, K. These letters are associated with the axes as follows:

(I) for the X, U, A axes (J) for the Y, V, B axes (K) for the Z, W, C axes

Example of a circular interpolation in the YZ and movement of the X axis with a helical pitch of 5 mm.

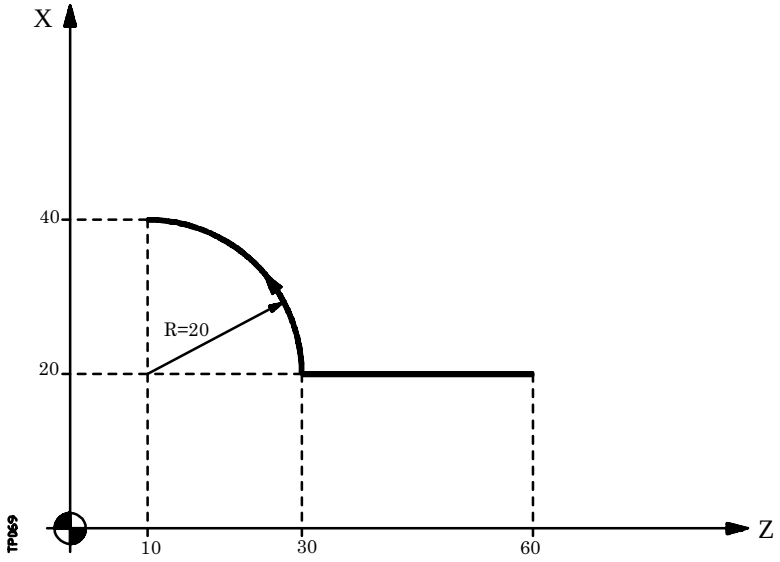
G02 Y Z J K X I5

6.8 TANGENTIAL ENTRY AT THE BEGINNING OF A MACHINING OPERATION (G37)

Via function **G37** you can tangentially link two paths without having to calculate the intersection points.

Function G37 is not modal, so it should always be programmed if you wish to start a machining operation with tangential entry:

Example with X axis programmed in radius:

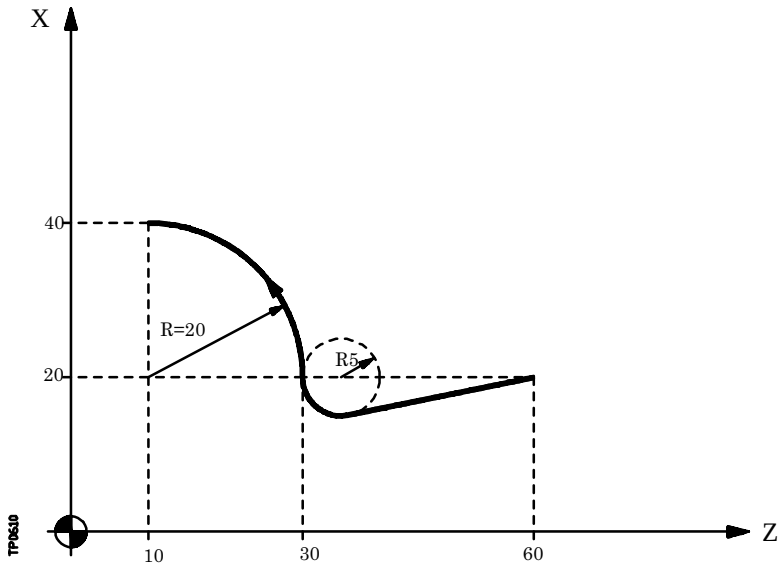


If the starting point is X20 Z60 and you wish to machine an arc (the path of approach being straight) you should program:

```
G90  G01  X20   Z30
     G03  X40   Z10   R20
```

If, however, in the same example you require the entrance of the tool to the part to be machined tangential to the path and describing a radius of 5 mm, you should program:

```
G90  G01  G37  R5    X20  Z30
     G03  X40  Z10   R20
```



As can be seen in the figure, the CNC modifies the path so that the tool starts to machine with a tangential entry to the part.

You have to program Function **G37** plus value **R** in the block which includes the path you want to modify.

R5.5 should appear in all cases following G37, indicating the radius of the arc which the CNC enters to obtain tangential entry to the part. Its value must always be positive.

Function G37 should only be programmed in the block which includes a straight-line movement (G00 or G01). If you program in a block which includes circular movement (G02 or G03), the CNC displays the corresponding error.

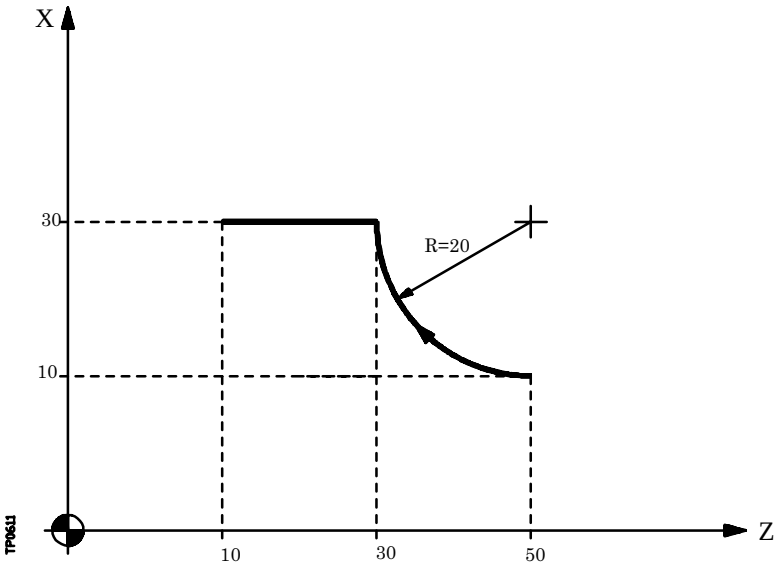
6.9 TANGENTIAL EXIT AT THE END OF A MACHINING OPERATION (G38)

Function **G38** enables the ending of a machining operation with a tangential exit of the tool. The path should be in a straight line (G00 or G01). Otherwise, the CNC will display the corresponding error.

Function G38 is not modal, so it should be programmed whenever a tangential exit of the tool is required.

Value **R 5.5** should always appear after G38. It also indicates the radius of the arc which the CNC applies to get a tangential exit from the part. This R value must always be positive.

Example with X axis programmed in radius:

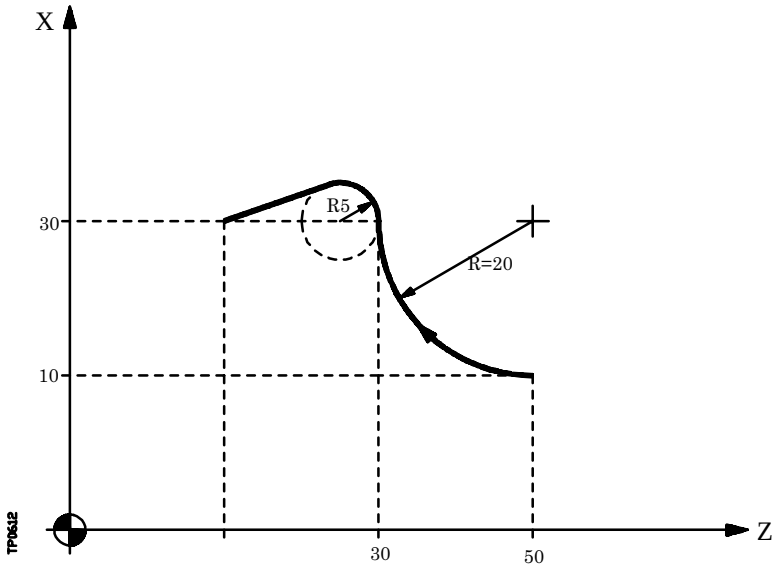


If the starting point is X10 Z50 and you wish to machine an arc (with the approach and exit paths in a straight line), you should program :

```
G90  G02  X30  Z30  R20
     G01  X30  Z10
```

If, however, in the same example you wish the exit from machining to be done tangentially and describing a radius of 5 mm, you should program :

```
G90  G02  G38  R5    X30  Z30  R20
     G01  X30  Z10
```



6.10 AUTOMATIC RADIUS BLEND (G36)

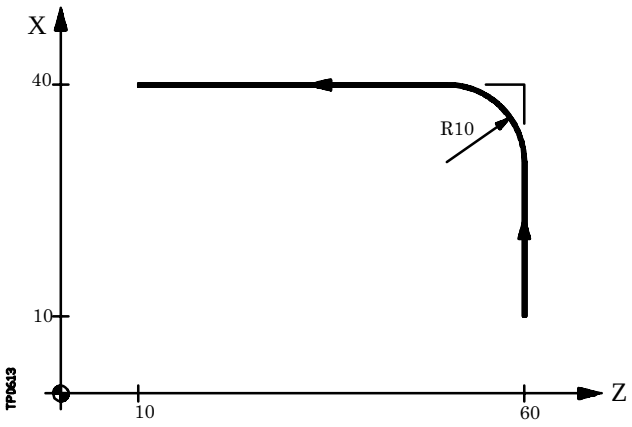
With function G36, it is possible to round a corner with a specific radius, without having to calculate the center nor the starting and end points of the arc.

Function G36 is not modal, so it should be programmed whenever controlled corner rounding is required.

This function should be programmed in the block in which the movement the end you want to round is defined.

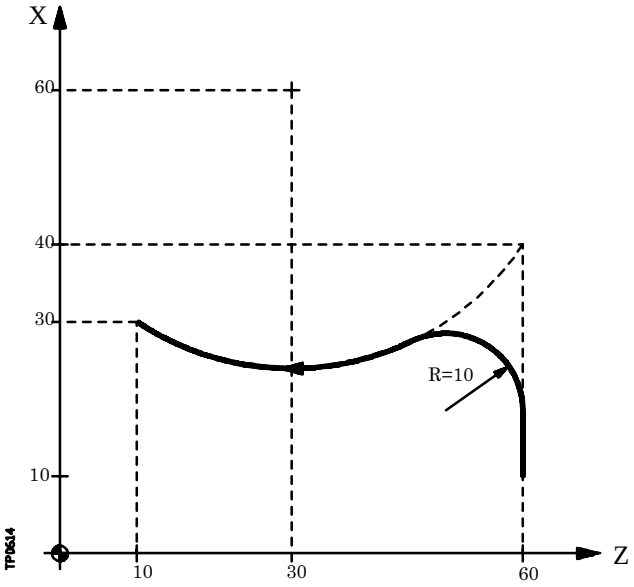
The **R5.5** value should always follow G36. It also indicates the rounding radius which the CNC applies to get the required corner rounding. This R value must always be positive.

Examples with X axis programmed in diameter:

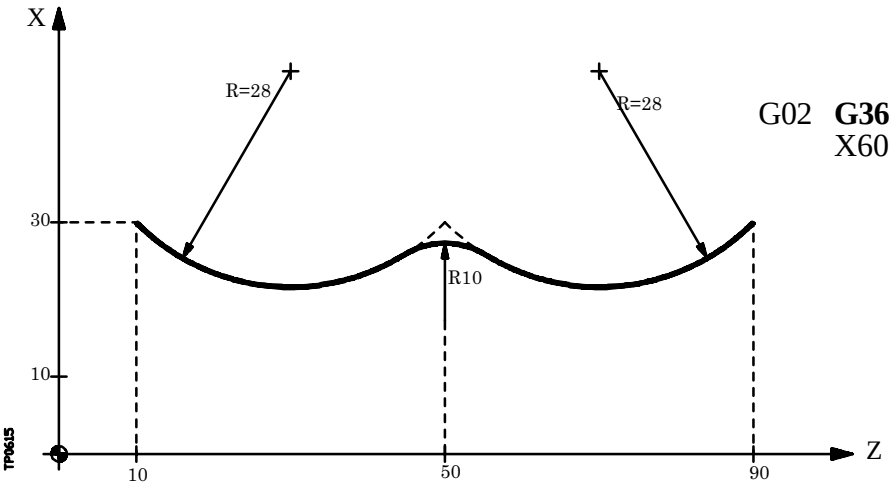


G01 **G36 R10** X80
Z10

G90 X60 Z90




```
G90  X20  Z60
G01  G36  R10  X80
G02  X60  Z10  I20    K-30
```




```
G02  G36  R10  X60 Z50 R28
     X60 Z10 R28
```

6.11 AUTOMATIC CHAMFER BLEND (G39)

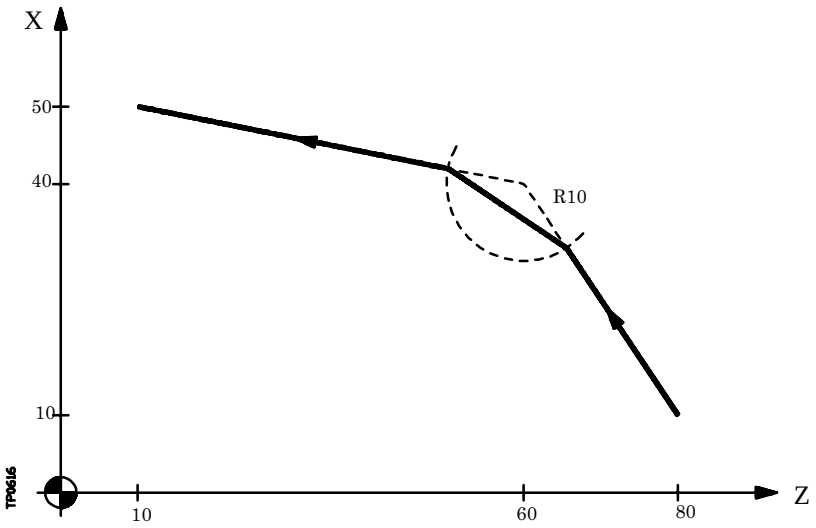
In machining operations it is possible (using G39) to chamfer corners between two straight lines, without having to calculate intersection points.

Function G39 is not modal, so it should be programmed whenever the chamfering of a corner is required.

This function should be programmed in the block in which the movement whose end you want to chamfer is defined.

The **R5.5** value should always follow G39. It also indicates the distance from the end of the programmed movement as far as the point where you wish to carry out the chamfering. This R value must always be positive.

Example with X axis programmed in diameter:



G90 X20 Z80
G01 **G39** **R10** X80 Z60
X100 Z10

6.12 ELECTRONIC THREAD CUTTING (G33)

If the spindle has a rotary encoder, it is possible to make threads with a cutter by means of function **G33**.

Although this threading is often done along the entire length of an axis, the CNC enables threading to be done interpolating more than one axis at a time.

Programming format:

G33 X.....C L Q

X...C ±5.5 End point of the thread

L5.5 Thread pitch

Q ±3.5 Optional. It indicates the spindle angular position (±359.9999) corresponding to the thread's starting point. With this, it is possible to make multiple entry threads.

If not programmed, a value of "0" is assumed.

Considerations:

When executing function G33 and before carrying out the electronic threading, the CNC performs a home search on the spindle and orients it to the angular position set by parameter Q.

Parameter "Q" is available when spindle machine parameter "M19TYPE=1".

When working in round corner (G05), several threads may be joined seamless on the same part.

When joining several threads, only the first one can have an entry angle (Q).

Neither the programmed feedrate "F" nor the spindle speed can be changed while function G33 is active. They both set at 100%.

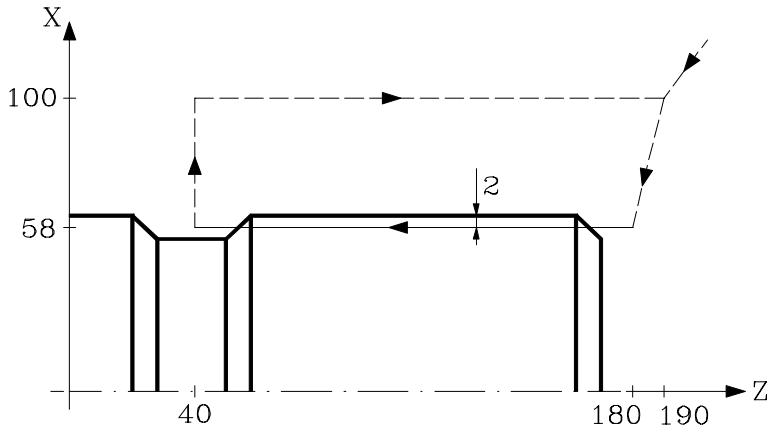
Function G33 is modal and incompatible with G00, G01, G02, G03 and G75.

On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC assumes G00 or G01 depending on the setting of general machine parameter "IMOVE".

Example when programming the X axis in diameter:

a.- Longitudinal threading

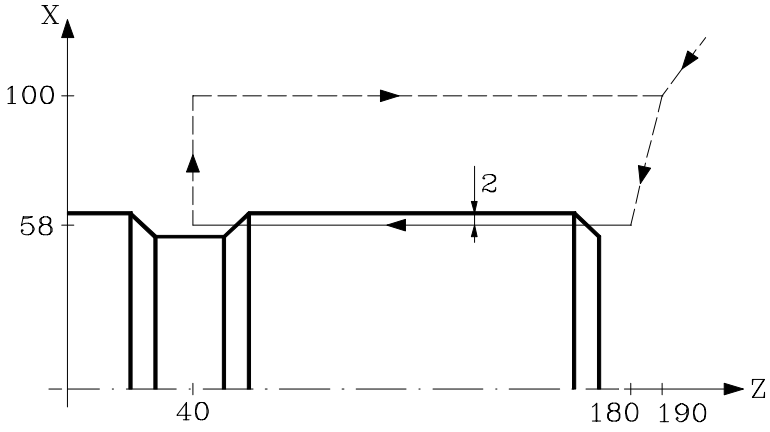
We want to make in a single pass a 5-mm pitch thread 2 mm deep.




```
G90  G00  X200   Z190
          X116   Z180
          G33  Z40    L5     ; Threading
          G00  X200
               Z190
```

b.- Multiple longitudinal threading.

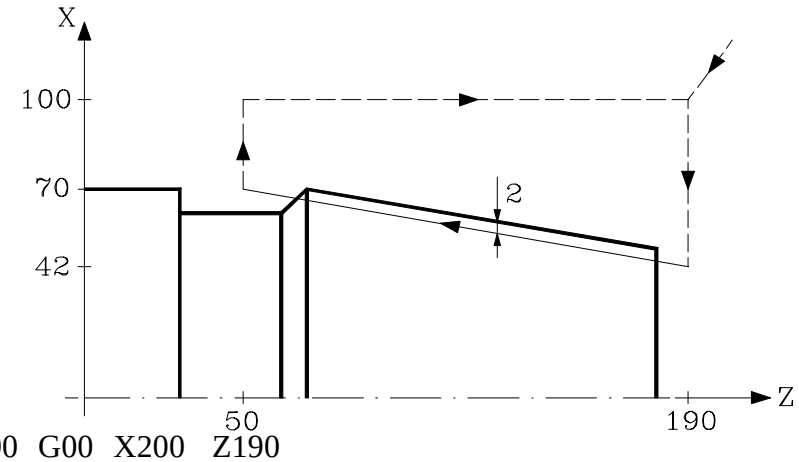
We want to make a dual entry thread. The threads are shifted 180º and each one is 2 mm deep and has a pitch of 5 mm.




```
G90  G00  X200   Z190
          X116   Z180
          G33  Z40    L5     Q0     ; 1st thread
          G00  X200
               Z190
               X116   Z180
          G33  Z40    L5     Q180 ; 2nd thread
          G00  X200
               Z190
```

c.- Tapered thread.

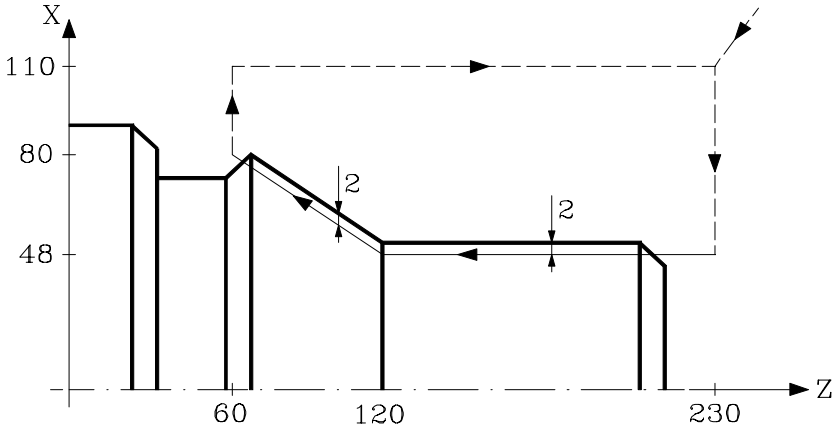
We wand to make in a single pass, a tapered thread 2 mm deep and a pitch of 5 mm.




```
G90  G00  X200   Z190
          X84
          G33  X140   Z50    L5     ; Thread
          G00  X200
               Z190
```

d.- Thread blending.

We want to blend a longitudinal thread and a tapered one 2 mm deep and with a 5 mm pitch.




```
G90  G05  G00  X220  Z230
               X96
               G33  Z120  L5          ; Thread
                    X160  Z60   L5     ; Thread
               G00  X220
                    Z230
```

6.13 ACTIVATE "C" AXIS (G15)

Preparatory function **G15** activates the **C** axis making it possible to machine on the cylindrical surface of part or on its front.

This is an optional feature and it requires the software option for 4 or more axes.

Inb order to use this G function, it is necessary to set one of the axes of the machine as a C axis (general machine parameters P0 thru P7) and define it as a normal rotary axis (machine parameter "**AXISTYPE**" for the **C** axis.

When the C axis is activated by means of G15, the CNC performs a home search if the spindle was previously operating in open loop.

The G15 function is modal and the logic output "**CAXIS**" (M5955) stays active while the C axis is active.

No more functions can be programmed in the block containing G15.

The C axis is deactivated returning to operate in regular spindle mode (open loop) after executing an M03, M04, M05, etc.

On the other hand, if machine parameter "PERCAX=NO" the CNC cancels the "C" function on power up or after executing an M02, M30, EMERGENCY or RESET.

6.13.1 MACHINING ON A CYLINDRICAL SURFACE

In order to machine on a cylindrical surface, first define the radius of the work surface by means of function **G15 R** and then select the main plane by means of **G16 Z C.**

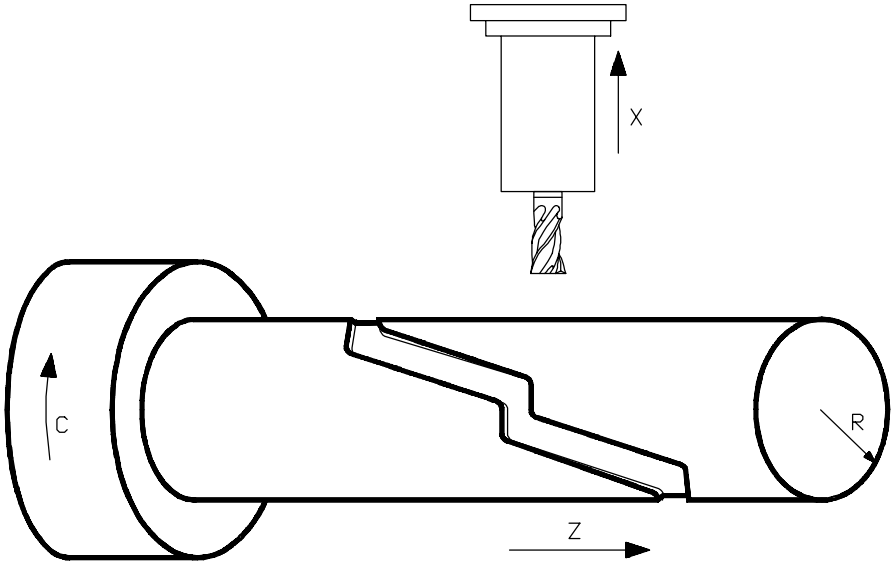
Next, program the profile to be machined with linear and circular interpolations as well as tool radius compensation, if so required.

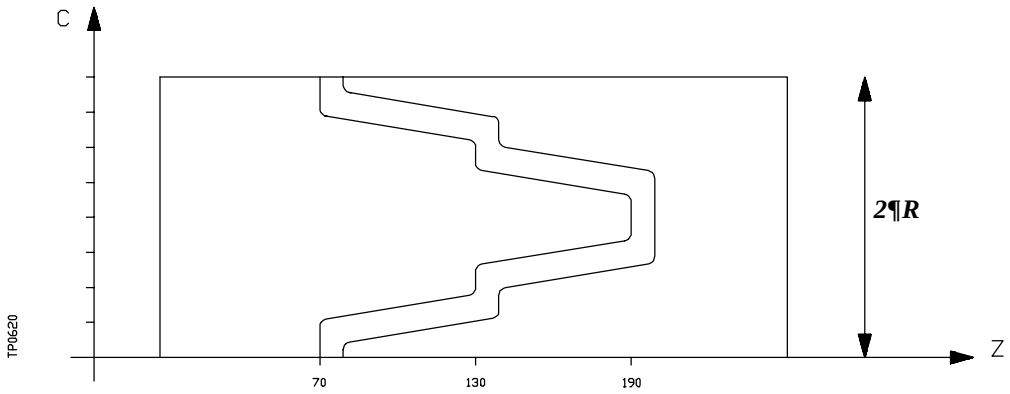
Porgram the C axis as a normal linear axis since the CNC will calculate the corresponding angular movements based on the radius value given by **G15 R.**

Program **G15 R** again to modify the radius at any point of the program.

Programming example, with the X axis in diameter, assuming that the work radius of the surface to be machined is R20.

```
G15 R20                    ; Machining on a cylindrical surface
```





G16 ZC

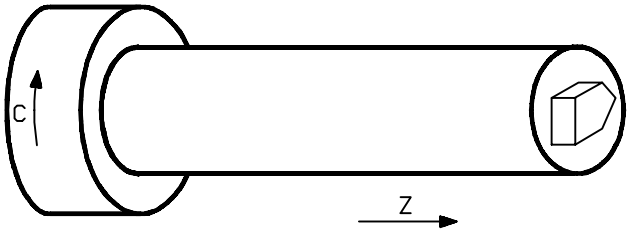
```
G90 G42 G01 Z70 C0 ; Positioning at starting point
G91 X-4                 ; Penetration
G90 G36 R3 C15.708
G36 R3 Z130 C31.416
G36 R3 C39.270
G36 R3 Z190 C54.978
G36 R3 C70.686
G36 R3 Z130 C86.394
G36 R3 C94.248
G36 R3 Z70 C109.956
G36 R3 C125.664
G91 X4                  ; Withdrawal
M30
```

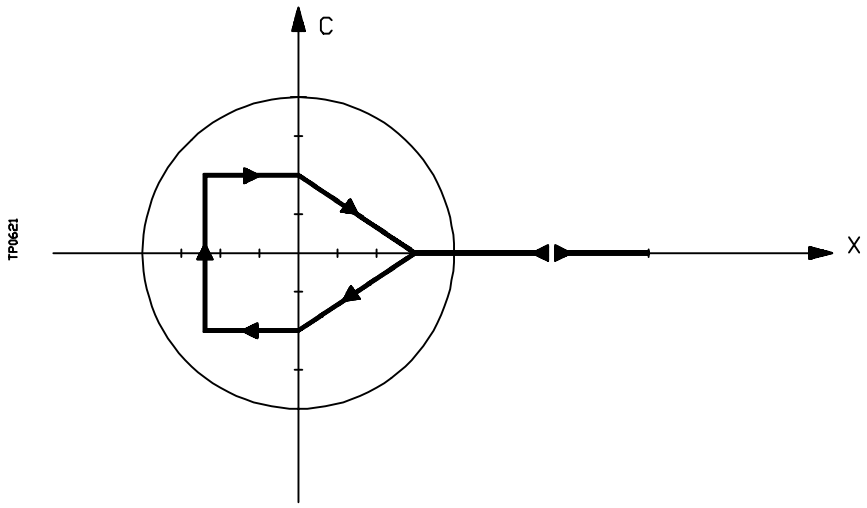
6.13.2 FACE MACHINING

In order to machine on the face of the part, first define function **G15** and then select the main plane by means of function **G16 X C.**

Next, program the profile to be machined with linear and circular interpolations as well as tool radius compensation, if so required.

Program the C axis as a normal linear axis and the values assigned to the X axis will be considered in radius regardless of the setting of machine parameter "**DFORMAT**" for the **X** axis.






```
G15                     ; Face machining
G16 XC
G0 X30                  ; Positioning at starting point
G0 Z-2                  ; Penetration
G1 G90 X15 F1000
X0 C-10
X-12
C10
X0
X15 C0
X30
G0 Z20
M30                     ; Withdrawal
```

6.14 MOVE TO HARDSTOP (G52)

By means of function G52 it is possible to program the movement of an axis until running into an object. This feature may be interesting for forming machines, live tailstocks, bar feeders, etc.

Its programming format is: **G52 X..C ±5.5**

After G52, program the desired axis as well as the target coordinate of the move.

The axis will move towards the programmed target coordinate until running into something. If the axis reaches the programmed target coordinate without running into the hardstop, it will stop.

Function G52 is not modal; therefore, it must be programmed every time this operation is to be carried out.

Also, it assumes functions G01 and G40 modifying the program history.
It is incompatible with functions G00, G02, G03, G41, G42, G75 and G76.

6.15 FEEDRATE "F" AS AN INVERTED FUNCTION OF TIME (G32)

There are instances when it is easier to define the time required by the various axes of the machine to reach the target point instead of defining a common feedrate for all of them.

A typical case may be when a linear axis (X, Y, Z) has to move together (interpolated) with a rotary axis programmed in degrees.

Function G32 indicates that the "F" functions programmed next set the time it takes to reach the target point.

In order for a greater value of "F" to indicate a greater feedrate, the value assigned to "F" is defined as "Inverted function of time" and it is assumed as the activation of this feature.

"F" units: 1/min

Example: G32 X22 F4 indicates that the movement must be executed in ¼ minute. That is, in 0.25 minutes.

Function G32 is modal and incompatible with G94 and G95.

On power-up, after executing M02, M30 or after an Emergency or Reset, the CNC assumes G94 or G95 depending on the setting of general machine parameter "IFFED".

Considerations:

The CNC variable PROGFIN will show the feedrate programmed as an inverted function of time and variable FEED will show the resulting feedrate in mm/min or inches/min.

If the resulting feedrate of any axis exceeds the maximum value set by machine parameter "MAXFEED", the CNC will apply this maximum value.

The programmed "F" is ignored on G00 movements. All the movements will be carried out at the feedrate set by axis machine parameter "G00FEED".

When programming "F0" the movement will be carried out at the feedrate set by axis machine parameter “MAXFEED”.

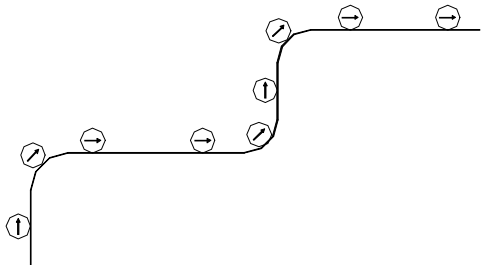
Function G32 may be programmed and executed in the PLC channel.

Function G32 is canceled in JOG mode.

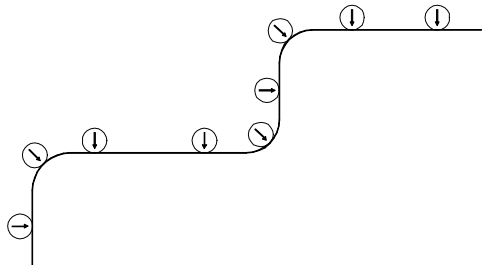
G32 is canceled when tracing. If it is programmed while tracing is active, the CNC will issue an error message.

6.16 TANGENTIAL CONTROL (G45)

With the "Tangential control" feature, the axis may maintain the same orientation with respect to the programmed path.



Orientation parallel to the path

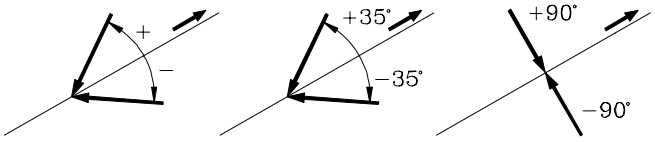


Orientation perpendicular to the path

The path is defined by the axes of the active plane. The axis maintaining the orientation must be a rotary rollover axis (A, B or C).

Programming format: **G45 Axis Angle**

Axis axis maintaining the orientation (A, B or C)
Angle Indicates the angular position in degrees with respect to the path (±359.9999). If not programmed, "0" will be assumed.



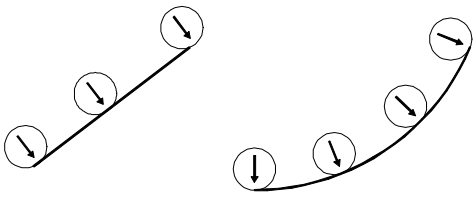
To cancel this function, program G45 alone (without defining the axis).

Every time G45 (tangential control) is activated, the CNC acts as follows:

1.- Positions the tangential axis, with respect to the first section in the programmed position.

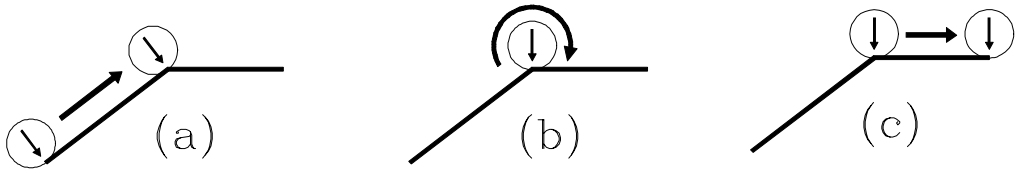
2.- The interpolation of the axes in the plane starts once the tangential axis has been positioned.

3.- On linear sections, the orientation of the tangential axis is maintained and in circular interpolations, the programmed orientation is maintained for the whole path.



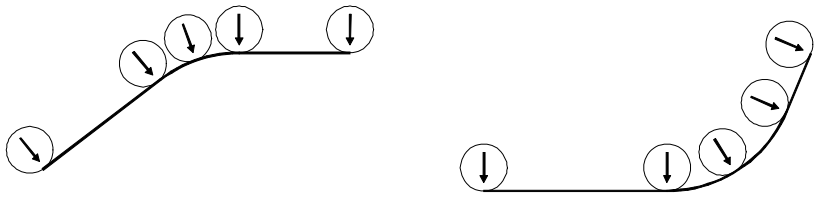
4.- If the joint of sections requires a new orientation of the tangential axis, the following takes place:

a) ends the current section.
b) orients the tangential axis with respect to the next section.
c) resumes execution.



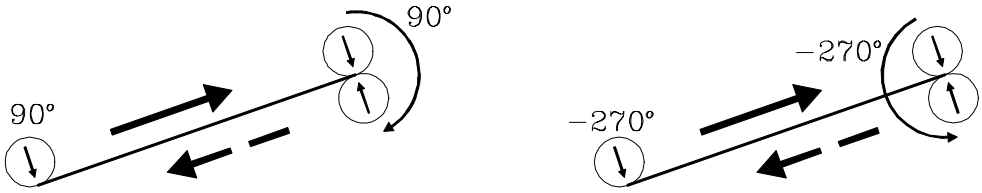
When working in round corner (G05), the tool orientation is not maintained at the corners since it begins before ending the current section.

It is recommended to work in square corner (G07). However, to work in round corner (G05), function G36 (automatic radius blend) should be used in order to also maintain tool orientation at the corners.



5.- To cancel the tangential control function, program G45 alone (without defining the axis).

Even when the tangential axis takes the same orientation by programming 90° or -270°, the turning direction in a direction change depends on the programmed value.



6.16.1 CONSIDERATIONS ABOUT FUNCTION G45

Tangential control, G45, is optional. It can only be executed in the main channel and is compatible with:

- Tool radius and length compensation (G40, 41, 42, 43, 44)
- Mirror image (G10, 11, 12, 13 14)
- Gantry axes , including the gantry axis associated with the tangential rotary axis.

The maximum feedrate while orienting the tangential axis is defined by machine parameter MAXFEED for that axis.

While tangential control is active, tool inspection is also possible. When accessing tool inspection, the tangential control is deactivated, the axes are free and when quitting tool inspection, tangential control may be activated again.

While in JOG mode, tangential control may be activated in MDI mode and the axes may be moved by programming blocks in MDI.

Tangential control is canceled when jogging the axes with the jog keys (not in MDI). Once the movement is over, tangential control is recovered.

On the other hand, the following is NOT possible:

- To define as tangential axis, one of the plane axes, the longitudinal axis or any other axis which is not rotary.
- To jog the tangential axis in JOG mode or by program using another G code while tangential control is active.
- Incline planes.

The TANGAN variable is read-only, from the CNC, PLC and DNC, associated with function G45. It indicates the angular position, in degrees, referred to the programmed path.

Also, general logic output TANGACT (M5558) indicates to the PLC that function G45 is active.

Function G45 is modal and is canceled when executing G45 alone (without defining the

7. ADDITIONAL PREPARATORY FUNCTIONS

7.1 INTERRUPTION OF BLOCK PREPARATION (G04)

The CNC reads up to 20 blocks ahead of the one it is executing, with the aim of calculating beforehand the path to be followed.

Each block is evaluated (in its absence) at the time it is read, but if you wish to evaluate it at the time of execution of the block you use function **G04**.

This function holds up the preparation of blocks and waits for the block in question to be executed in order to start the preparation of blocks once more.

A case in point is the evaluation of the "status of block-skip inputs" which is defined in the block header.

Example:

```
        .
        .
        .
    G04                 ; interrupts block preparation
/1  G01   X10 Z20       ; block-skip condition "/1"
        .
        .
```

Function G04 is not modal, so it should be programmed whenever you wish to interrupt block preparation.

It should be programmed on its own and in the block previous to the one in which the evaluation in execution is required. Function G04 can be programmed as G4.

Every time G04 is programmed, active radius and length compensation are cancelled.

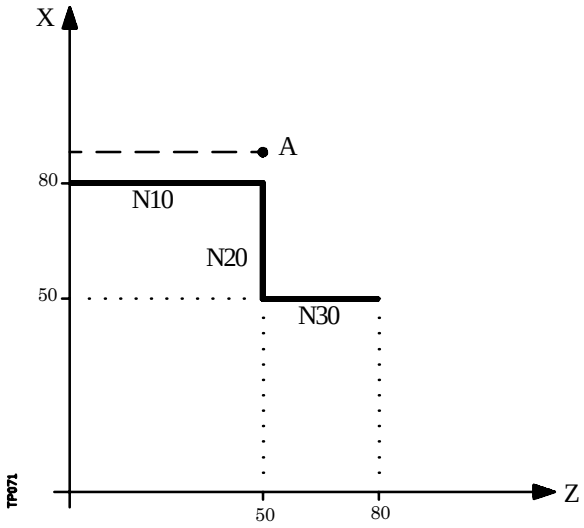
For this reason, care needs to be taken when using this function, because if it is introduced between machining blocks which work with compensation, unwanted profiles may be produced.

Example :

The following program blocks are executed in a section with G41 compensation :

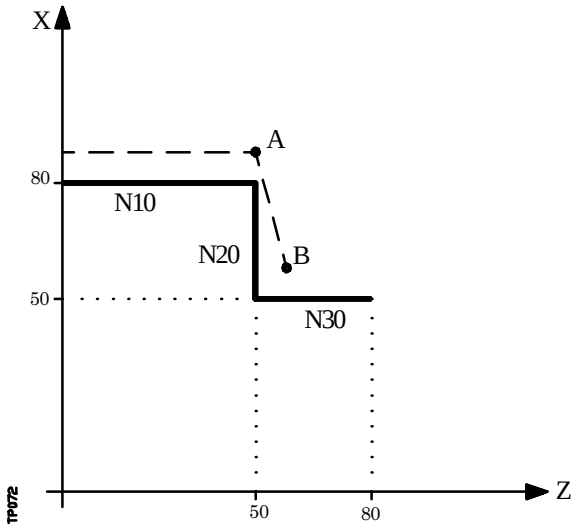
```
      .....
      .....
      N10  X80  Z50
      N15  G04
/1    N17  M10
      N20  X50  Z50
      N30  X50  Z80
      .....
      .....
```

Block N15 holds back the preparation of blocks so that the execution of block N10 ends up at point A.



Once the execution of block N15 has been carried out, the CNC continues preparing blocks starting from block N17.

Given that the next point corresponding to the compensated path is point “B”, the CNC moves the tool to this point, executing path “A-B”.



As you can see, the resulting path is not the required one, so we recommend avoiding the use of function G04 in sections which work with compensation.

7.2 DWELL (G04 K)

Timing can be programmed via function **G04 K**.

The timing value is programmed in hundredths of a second via format **K5 (0..99999).**

Example :

```
G04 K50   ;  Timing of 50 hundredths of a second (0.5 seconds)
G04 K200  ;  Timing of 200 hundredths of a second (2 seconds)
```

Function G04 K is not modal, so it should be programmed whenever timing is required. Function G04 K can be programmed as G4 K.

Timing is executed at the beginning of the block in which it is programmed.

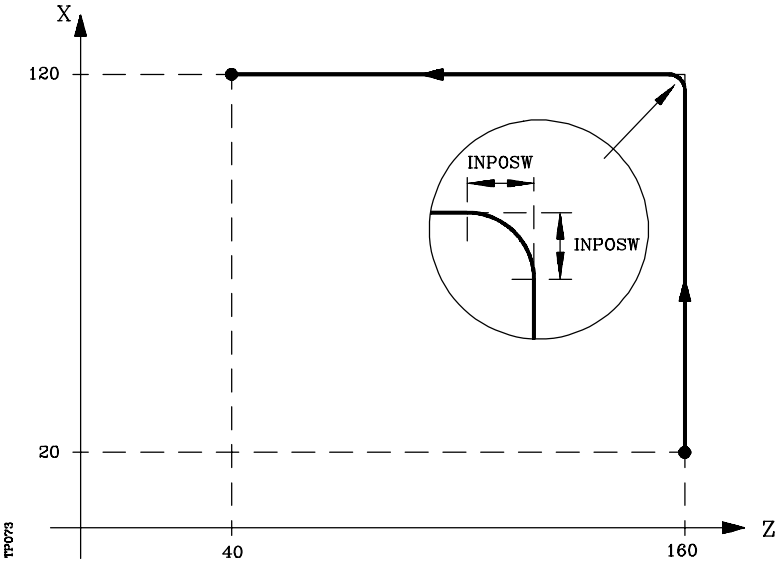
7.3 WORKING WITH SQUARE (G07) AND ROUND (G05,G50)CORNERS

7.3.1 SQUARE CORNER (G07)

When working in **G07** (square corner) the CNC does not start executing the following program block until the position programmed in the current block has been reached.

The CNC considers that the programmed position has been reached when the axis is within the "INPOSW" (in-position zone or dead band) from the programmed position.

Example:



G91 G01 **G07** X100 F100
Z-120

The theoretical and real profile coincide, obtaining square corners, as seen in the figure.

Function G07 is modal and incompatible with G05, G50 and G51. Function G07 can be programmed as G7.

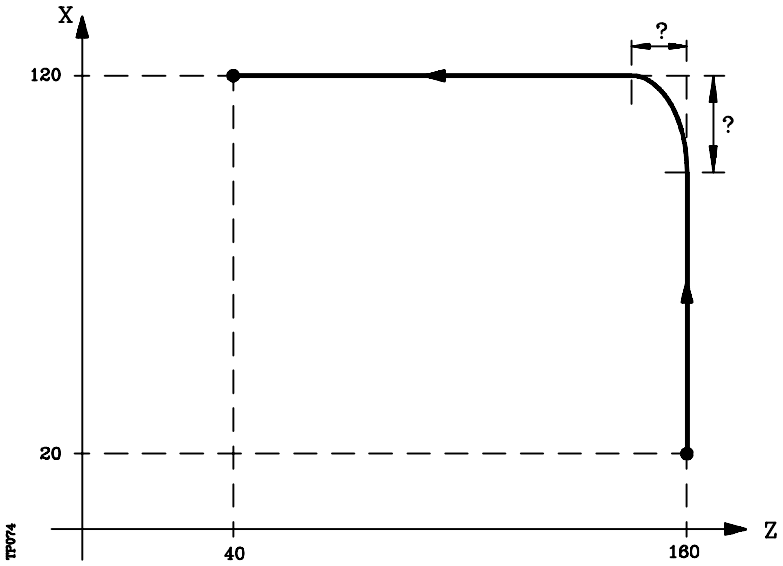
On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter "ICORNER" is set.

7.3.2 ROUND CORNER (G05)

When working in **G05** (round corner), the CNC starts executing the following block of the program as soon as the theoretical interpolation of the current block has concluded. It does not wait for the axes to physically reach the programmed position.

The distance prior to the programmed position where the CNC starts executing the next block depends on the actual axis feedrate.

Example :



G91 G01 **G05** X100 F100
Z-120

Via this function round corners can be obtained, as shown in the figure.

The difference between the theoretical and real profiles depends on the programmed feedrate value "F". The higher the feedrate, the greater the difference between both profiles.

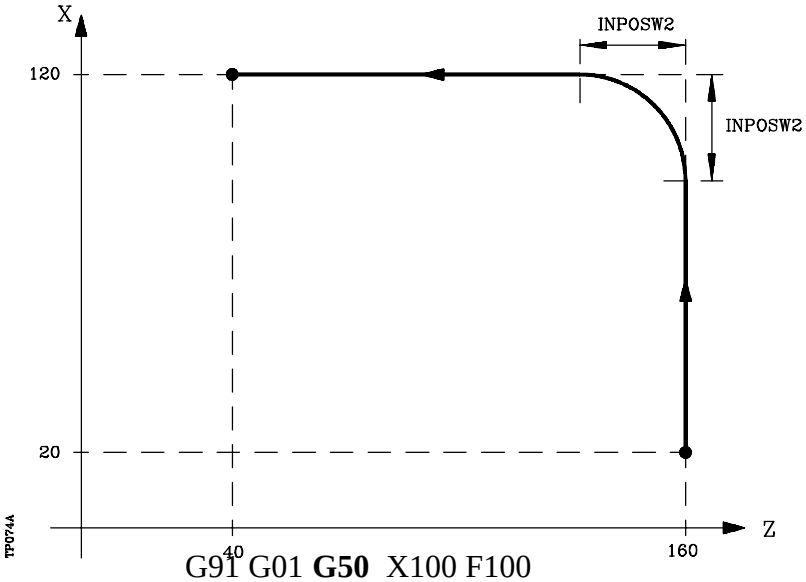
Function G05 is modal and incompatible with G07, G50 and G51. Function G05 can be programmed as G5.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter "ICORNER" is set.

7.3.3 CONTROLLED ROUND CORNER (G50)

When working in **G50** (controlled round corner); once the theoretical interpolation of the current block has concluded, the CNC waits for the axis to enter the area defined by machine parameter "INPOSW2" and it then starts executing the following block of the program.

Example :



G91 G01 **G50** X100 F100
Z-120

Function G50 assures that the difference between the theoretical and actual paths stays smaller than what was set by machine parameter "INPOSW2".

On the other hand, when working in G05, the difference between the theoretical and real profiles depends on the programmed feedrate value “F”. The higher the feedrate, the greater the difference between both paths.

Function G50 is modal and incompatible with G05, G07 and G51.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter “ICORNER” is set.

7.4 LOOK-AHEAD (G51)

Usually, a program consisting of very small movement blocks (CAM, digitizing, etc.) run very slowly.

With this feature, high speed machining is possible for this type of programs.

It is recommended to have the CPU-TURBO feature when using LOOK-AHEAD because the CNC has to analyze the machining path ahead of time (up to 50 blocks) in order to calculate the maximum feedrate for each section of the path.

The programming format is: G51 [A] E

A (0-255) Is optional and it defines the percentage of acceleration to be applied. When not programmed or programmed with a "0" value, the CNC assumes the acceleration value set by machine parameter for each axis.
E (5.5) Maximum contouring error allowed.

Parameter "A" permits using a standard working acceleration and another one to be used when executing with Look-Ahead.

The smaller the "E" parameter value, the lower the machining feedrate.

When operating with "Look-Ahead", it is a good idea to adjust the axes so their following error (lag) is as small as possible because the contouring error will be at least equal to the minimum following error.

When calculating the axis feedrate, the CNC takes into consideration the following aspects:

* The programmed feedrate.
* The curvature and the corners.
* The maximum feedrates of the axes.
* The maximum accelerations.

If any of the circumstances listed below occurs while executing with Look-Ahead, the CNC slows down to "0" at the previous block and it recovers the machining conditions for Look-Ahead in the next motion block.

* Motionless block.
* Execution of auxiliary functions (M, S, T).
* Single block execution mode.
* MDI mode.
* TOOL INSPECTION mode.

If a Cycle Stop, Feed-Hold, etc. occurs while executing in Look-Ahead mode, the machine may not stop at the current block, several additional blocks will be necessary to stop with the permitted deceleration.

Function G51 is modal and incompatible with G05, G07 and G50. Should any of them be programmed, function G51 will be canceled and the new one will be selected.

On the other hand, the CNC will issue Error 7 (Incompatible G functions) when programming any of the following functions while G51 is active:

* G23, G26, G27 Tracing
* G33 Electronic threading
* G52 Movement against hardstop
* G74 Home search
* G75, G76 Probing
* G95 Feedrate per revolution

Function G51 must be programmed alone in a block and there must be no more information in that block.

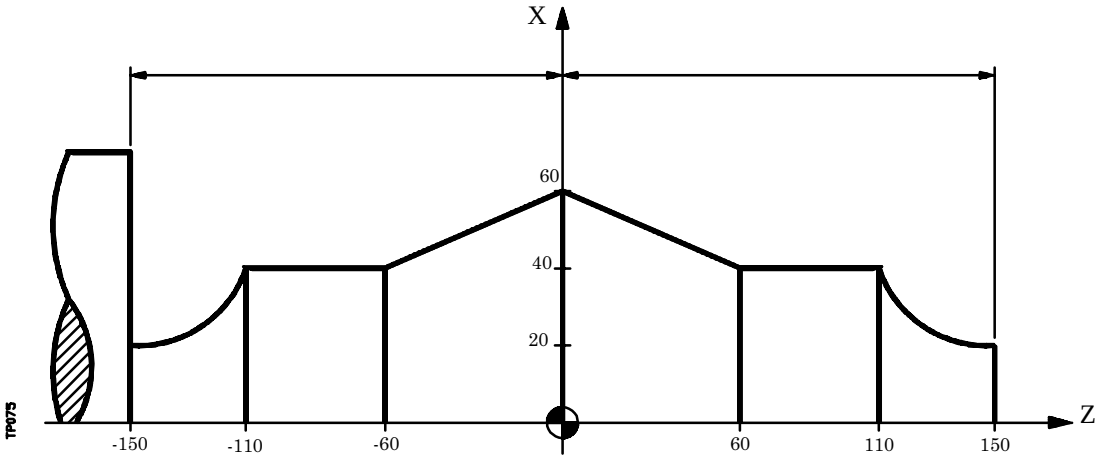
On power-up, after executing an M02, M30, of after an EMERGENCY or RESET, the CNC will cancel G51, if it was active, and it will assume G05 or G07 according to the setting of general machine parameter “ICORNER”.

7.5 MIRROR IMAGE (G10, G11. G12, G13, G14)

G10 : cancel mirror image
G11 : mirror image on X axis
G12 : mirror image on Y axis
G13 : mirror image on Z axis
G14 : mirror image on any axis (X..C), or in several at the same time.
Examples : G14 W G14 X Z A B

When the CNC works with mirror images, it executes the movements programmed in the axes which have mirror image selected, with the sign changed.

Example :



The following subroutine defines the machining of part “A”.

```
G90  G00  X40   Z150
     G02  X80   Z110  R60
     G01        Z60
          X120  Z0
```

The programming of all parts would be :

```
Execution of subroutine    ;   machines "A"
          G13              ;   mirror image on Z axis.
          M30              ;   end of program.
```

Functions G11, G12, G13, and G14 are modal and incompatible with G10.

G11, G12, and G13 can be programmed in the same block, because they are not incompatible with each other. Function G14 must be programmed alone in the block.

If while one of the mirror imaging functions (G11, G12, G13 or G14) is active, a new coordinate origin (part zero) is preset with G92, this new origin will not be affected by the mirror imaging function.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G10.

7.6 SCALING FACTOR (G72)

By using function **G72** you can enlarge or reduce programmed parts.

In this way, you can produce families of parts which are similar in shape but of different sizes with a single program.

Function G72 should be programmed on its own in a block. There are two formats for programming G72 :

Scaling factor applied to all axes.
Scaling factor applied to one or more axes.

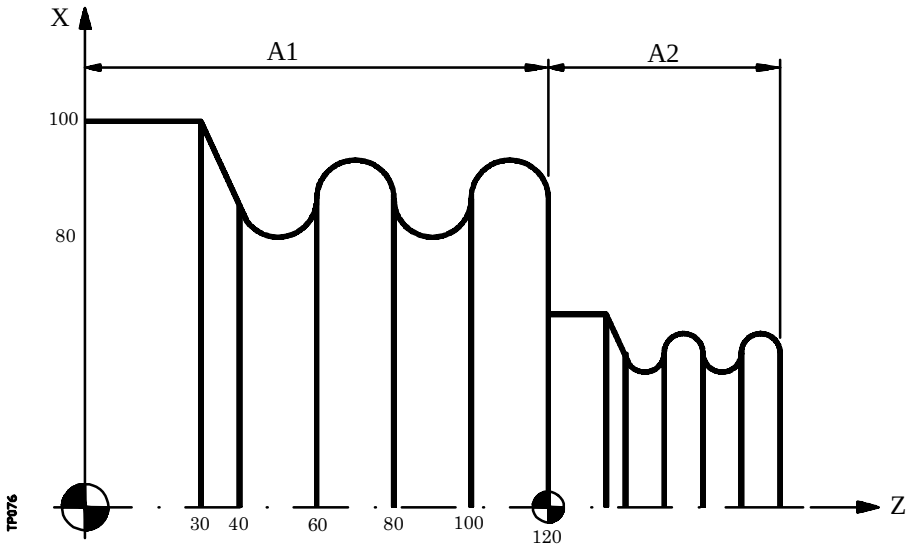
7.6.1 SCALING FACTOR APPLIED TO ALL AXES

The programming format is as follows :

G72 S5.5

Following G72 all coordinates programmed are multiplied by the value of the scaling factor defined by S until a new G72 scaling factor definition is read or the definition is cancelled.

Programming example with X axis in diameter:



The following subroutine defines the basic machining of the part.

```
G90  X200  Z0
G01  X200  Z30
G01  X160  Z40
G03  X160  Z60   I0  K10
G02  X160  Z80   I0  K10
G03  X160  Z100  I0  K10
G02  X160  Z120  I0  K10
```

The programming of the part would be :

```
Execution of subroutine   ;  machines "A1"
      G92 Z0              ;  coordinate preset
                             (zero offset)
      G72 S0.5            ;  applies scaling factor 0.5
Execution of subroutine   ;  machines "A2"
      G72 S1              ;  cancels scaling factor
      M30                 ;  end of program
```

7.6.2 SCALING FACTOR APPLIED TO ONE OR MORE AXES

The programming format is :

G72 X...C 5.5

After **G72** the axis or axes and the required scaling factor are programmed.

All blocks programmed after G72 are treated by the CNC as follows :

The CNC calculates the movement of all the axes in relation to the programmed path and compensation.

It then applies the scaling factor indicated to the calculated movement of the corresponding axis or axes.

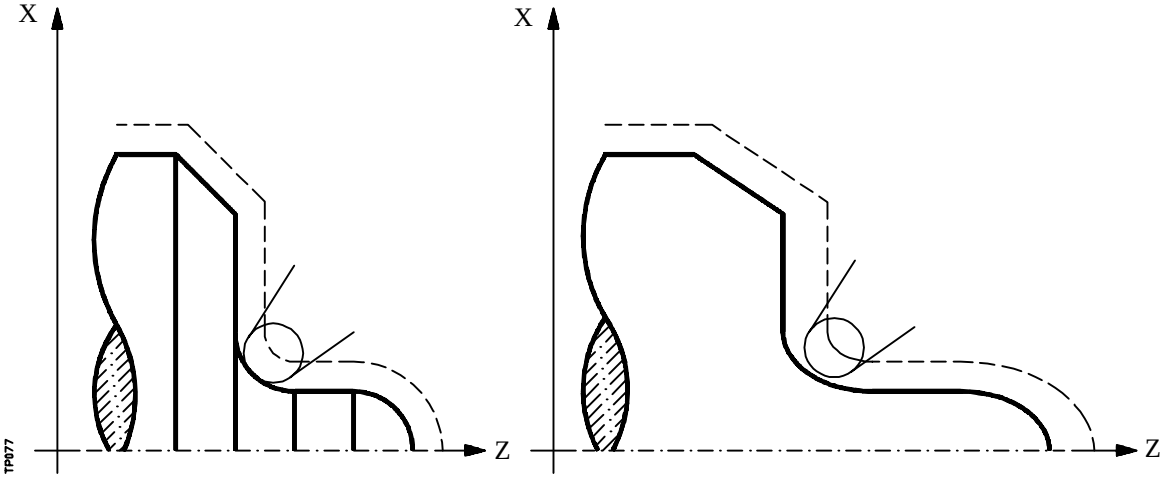
If the scaling factor is applied on one or more axes, the CNC will apply the scaling factor indicated both to the movement of the corresponding axis or axes and to their feedrate.

If, within the same program, both scaling factor types are applied, the one applied to all the axes and the one for one or several axes, the CNC applies a scaling factor equal to the product of the two scaling factors programmed for this axis to the axis or axes affected by both types.

Function G72 is modal and will be cancelled when the CNC is turned on, after executing M02, M30 or after an EMERGENCY or RESET.

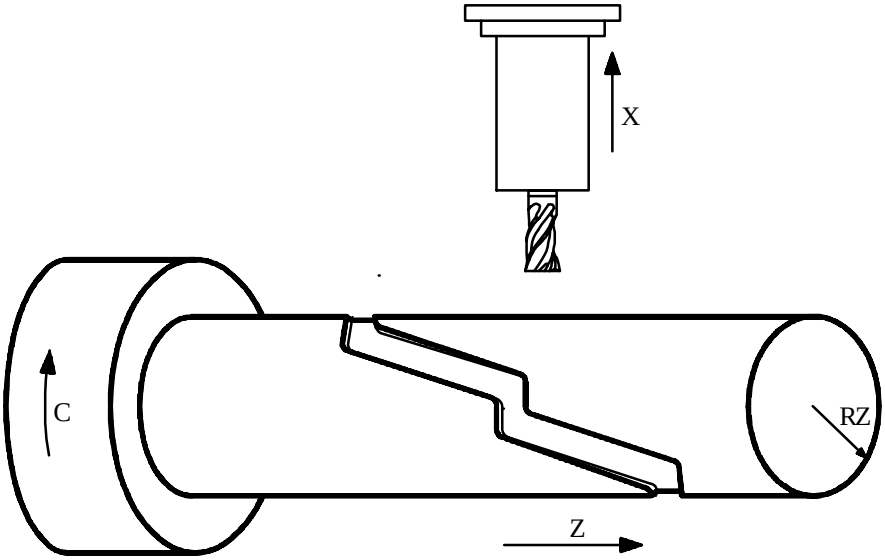
Examples:

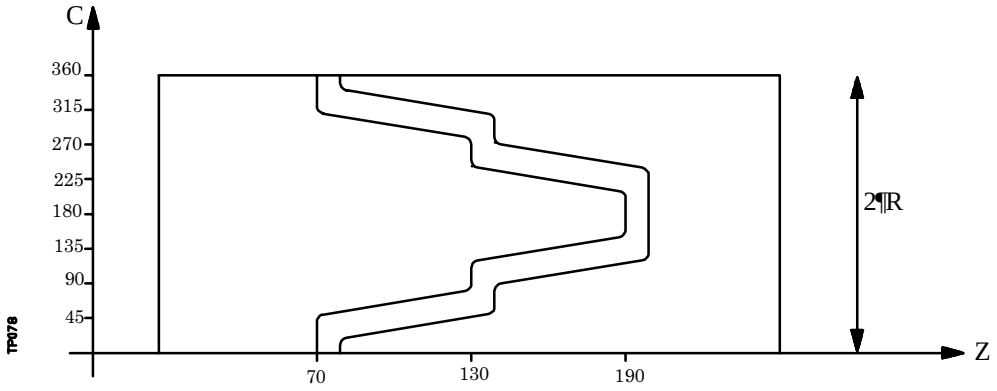
Application of the scaling factor to the Z axis, working with tool radius compensation.



As can be observed, the tool path does not coincide with the required path, as the scaling factor is applied to the calculated movement.

However, if a scaling factor equal to 360/(2¶R) is applied to a rotary axis, R being the radius of the cylinder on which you wish to machine, this axis can be considered linear, and any figure with tool radius compensation can be programmed on the cylindrical surface.





Programming example with the X axis in diameter assuming that the radius of the cylinder is R20:

Scaling Factor to be applied = 360/(2¶R) = 2.86

```
G16  ZC
G90  G42  G01   Z70   C0    ;  Position at initial point
G91  X-4                   ;  Penetration
G72  C2.86                 ;  Scaling Factor
G90  G36  R5    C45
G36  R5   Z130  C90
G36  R5   C112.5
G36  R5   Z190  C157.5
G36  R5   C202.5
G36  R5   Z130  C247.5
G36  R5   C270
G36  R5   Z70   C315
G36  R5   C360
G91  X4                    ;  Withdrawal
G72  C1                    ;  Cancel Scaling Factor
M30
```

7.7 SLAVED AXIS/CANCELLATION OF SLAVED AXIS

The CNC enables two or more axes to be coupled together. The movement of all axes is subordinated to the movement of the axis to which they were coupled.

There are three possible ways of coupling axes :

Mechanical coupling. This is imposed by the manufacturer of the machine, and is selected via the axis machine parameter “GANTRY”.

By means of the PLC. This enables the coupling and uncoupling of each axis through logic input on the CNC “SYNCHRO1”, “SYNCHRO2”, “SYNCHRO3”, “SYNCHRO4”, and “SYNCHRO5”. Each axis is coupled to the one indicated in the axis machine parameter “SYNCHRO”.

By means of the program. This enables electronic coupling and uncoupling between two or more axes, through functions G77 and G78.

7.7.1 SLAVED AXIS (G77)

Function G77 allows the selection of both the master axis and the slaved axis (axes). The programming format is as follows :

G77 < Axis 1 > < Axis 2 > < Axis 3 > < Axis 4 > < Axis 5>

In which < Axis 2 > < Axis 3 > < Axis 4 > < Axis 5> indicate the slave axes you wish to couple to the master axis < Axis 1 >. You have to define < Axis 1 > and < Axis 2 >, the programming of the rest of the axes being optional.

Example :

```
G77 X Y U    ;  slaves Y and U axes to X axis
```

The following rules should be observed when doing electronic axis slaving:

You may use one or two different electronic couplings.

```
G77 X Y U    ;  slaves Y and U axes to X axis
G77 V Z      ;  slaves Z axis to V axis
```

You cannot couple one axis to two others at the same time.

```
G77 V Y      ;  slaves Y axis to V axis
G77 X Y      ;  gives an error signal, because Y axis is slaved to V axis.
```

You can slave several axes to one in successive steps.

```
G77 X Z      ;  slaves Z axis to X axis
G77 X U      ;  slaves U axis to X axis —> Z U slaved to X
G77 X Y      ;  slaves Y axis to X axis —> Y Z U slaved to X
```

A pair of axes which are already coupled to each other cannot be slaved to another axis.

```
G77 Y U      ;  slaves U axis to Y axis
G77 X Y      ;  gives an error signal, because Y axis is slaved to U axis.
```

7.7.2 SLAVED AXIS CANCELLATION (G78)

Function G78 enables you to uncouple all the axes which are slaved, or only uncouple indicated axes.

G78 Uncouples all slaved axes.

G78 <Axis 1><Axis 2><Axis 3><Axis 4> Only uncouples indicated axes.

Example :

```
G77 X Y U   ; slaves Y and U axes to X axis
G77 V Z     ; slaves Z axis to V axis
G78 Y       ; uncouples Y axis, but U -> X and Z -> V stay
              slaved.
G78         ; uncouples all axes.
```

8. *TOOL COMPENSATION*

The CNC has a tool offset table, its number of components being defined via the general machine parameter “NTOFFSET”. The following is specified for each tool offset :

* Nominal tool length along X axis. It will be programmed in radius and in the units established by general machine parameter “INCHES” being its format X±5.5

* Nominal tool length along Z axis. It will be programmed in radius and in the units established by general machine parameter “INCHES” being its format Z±5.5

* The type of tool indicates the shape of the tool and how it has been calibrated. It is defined according to location codes F0 through F10.

  Location codes F0 and F9 must only be used when calibrating the tool tip center and not the sides of the cutter.

  When using an end-mill or drill, F10 must be selected.

  The following pages show how to use these location codes (F0 through F10).

* Nominal tool radius. It will be programmed in radius and in the units established by general machine parameter “INCHES” being its format R±5.5

* Tool length wear along X axis, in the units established by general machine parameter “INCHES” and in I±5.5 format. The CNC adds this value to the nominal tool length along X axis to calculate the real radius (X+I).

* Tool length wear along Z axis, in the units established by general machine parameter “INCHES” and in K±5.5 format. The CNC adds this value to the nominal tool length along X axis to calculate the real radius (Z+K).

When tool radius compensation is required (G41 or G42), the CNC takes into account the tool location code and it applies the R value of the selected tool offset as the compensation value.

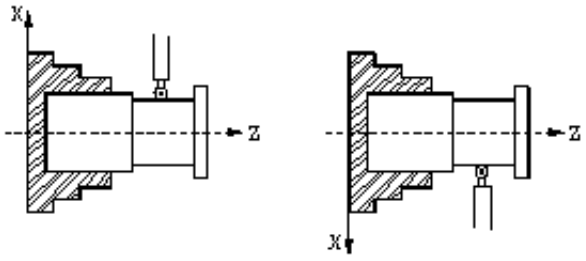
If no tool offset has been programmed, the CNC applies the tool offset **D0** with X=0, Z=0, F=0, R=0 I=0 and K=0.

Every time a tool is selected (active tool) and on the first X or Z axis move, the CNC applies automatically the tool length compensation corresponding to the selected tool offset . the sum of X+I along the X axis and the sum Z+K along the Z axis.

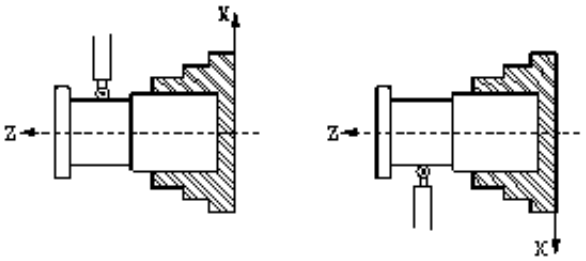
If there is no active tool, the CNC will apply no tool length compensation.

CODIGOS DE FORMA LOCATION CODES

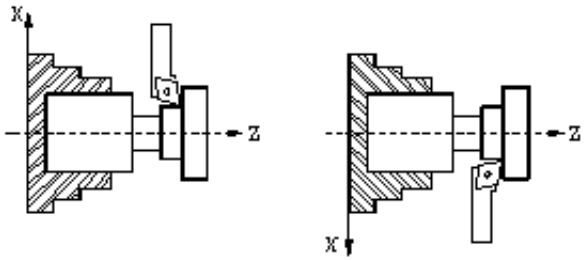
CODIGOS F0 y F9



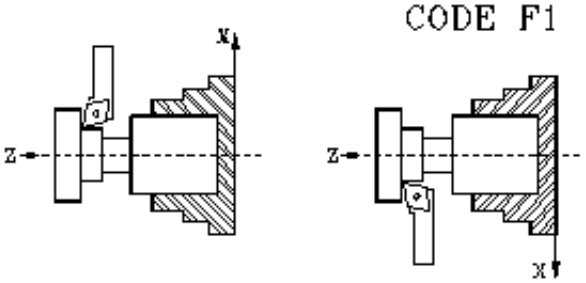
CODES F0 and F9



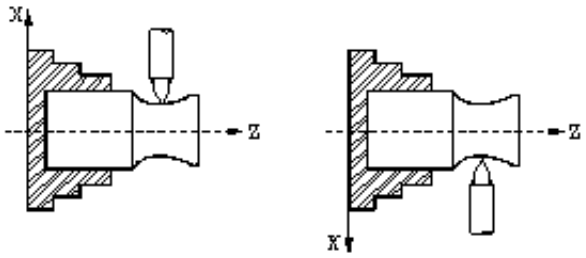
CODIGO F1



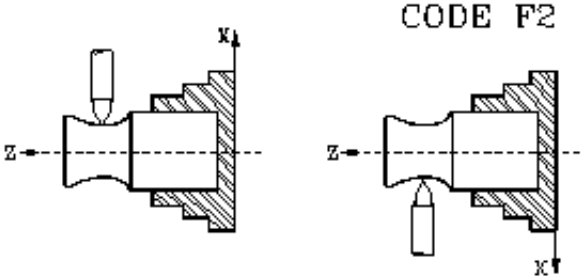
CODE F1



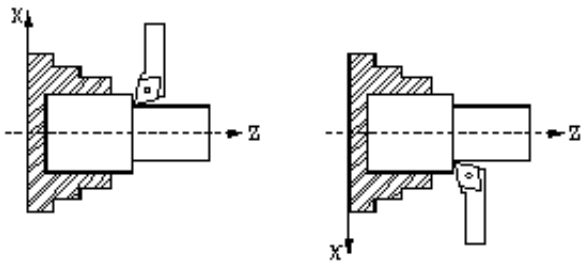
CODIGO F2



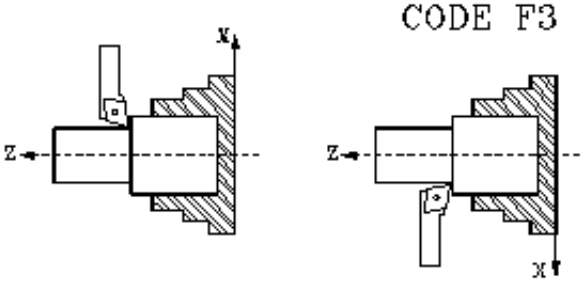
CODE F2



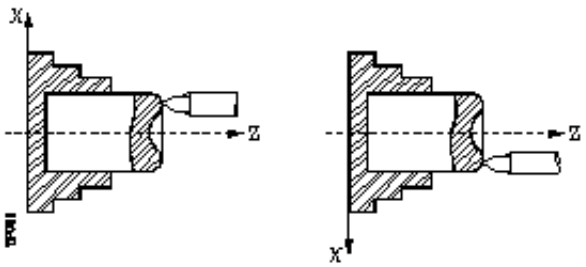
CODIGO F3



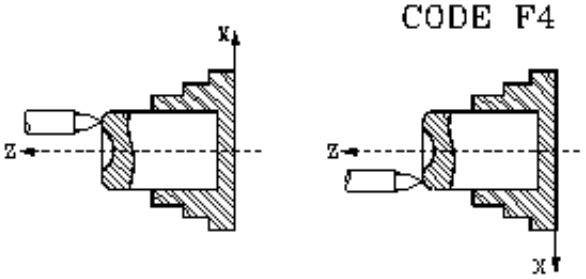
CODE F3



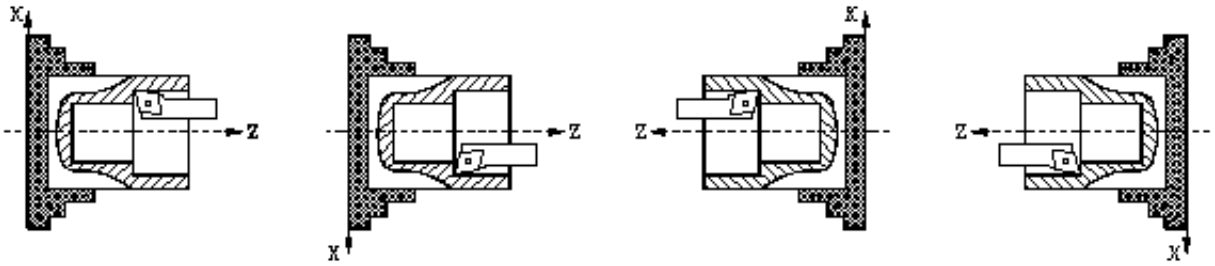
CODIGO F4



CODE F4

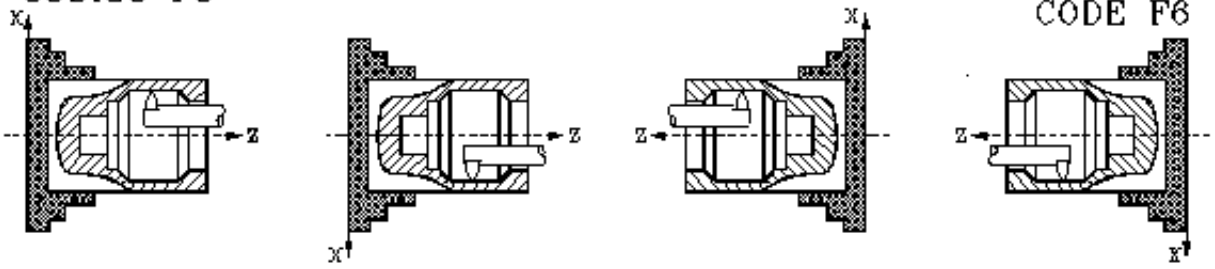


CODIGO F5



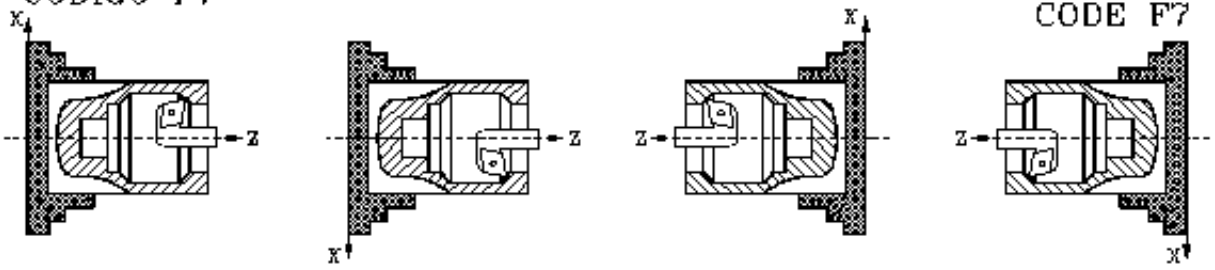
CODE F5

CODIGO F6



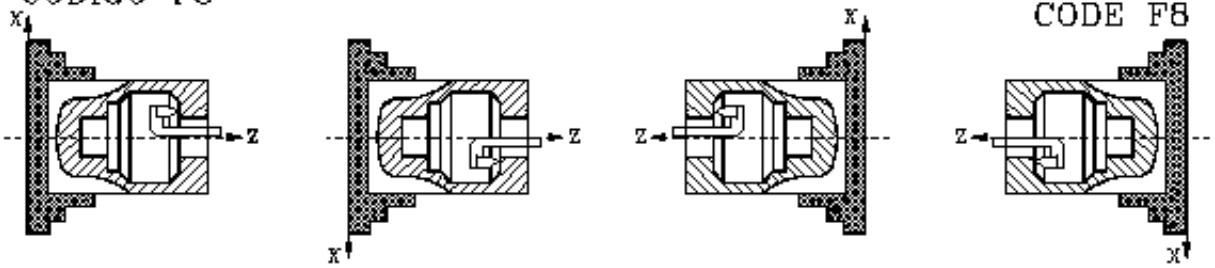
CODE F6

CODIGO F7



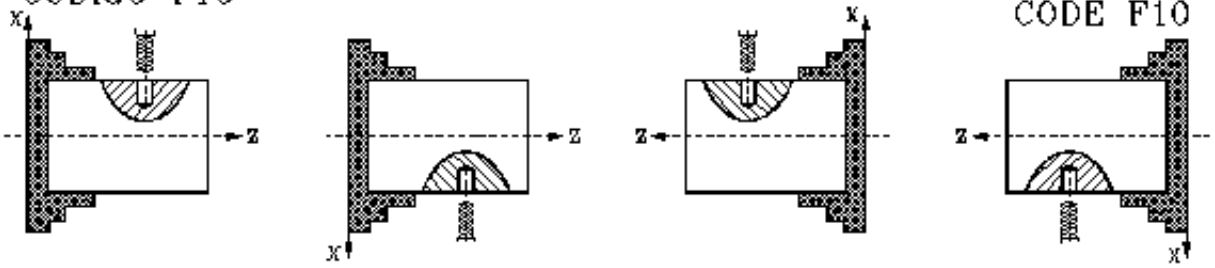
CODE F7

CODIGO F8



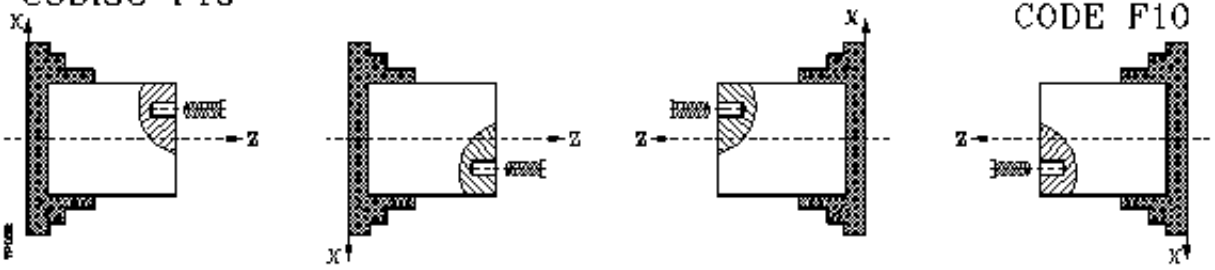
CODE F8

CODIGO F10



CODE F10

CODIGO F10



CODE F10

8.1 TOOL RADIUS COMPENSATION (G40, G41, G42)

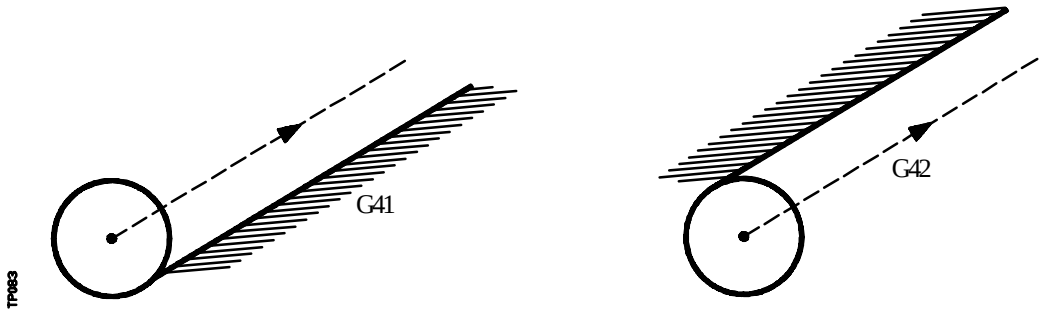
In normal machining operations, it is necessary to calculate and define the path of the tool taking its location code (F) and radius (R) into account so that the required dimensions of the part are achieved.

Tool radius compensation allows the direct programming of part contouring and of the tool radius without taking the dimensions of the tool into account.

The CNC automatically calculates the path the tool should follow based on the contour of the part, and the tool's location code and its radius value stored in the tool offset table corresponding to that tool.

There are three preparatory functions for tool radius compensation:

G40 Cancelling of tool radius compensation
G41 Tool radius compensation to the left of the part.
G42 Tool radius compensation to the right of the part.



G41. The tool is to the left of the part, depending on the machining direction.

G42. The tool is to the right of the part, depending on the machining direction.

Tool values **X, Z, F, R, I, K** should be stored in the tool offset table before starting machining, or should be loaded at the beginning of the program via assignments to variables **TOX, TOZ, TOF, TOR, TOI, TOK.**

Once the plane in which compensation will be applied has been chosen via codes G16, G17, G18, or G19, this is put into effect by **G41** or **G42**, assuming the value of the tool offset selected via code **D**, or (in its absence) by the tool offset shown in the tool table for the selected tool (**T**).

Functions G41 and G42 are modal and incompatible to each other. They are cancelled by G40, G04 (interruption of block preparation), G53 (programming with reference to machine zero), G74 (home search), machining canned cycles (G66, G68, G69, G81, G82, G83, G84, G85, G86, G87, G88, G89) and also on power-up, after executing M02, M30 or after EMERGENCY or RESET.

8.2 ACTIVATING TOOL RADIUS COMPENSATION

Once the plane in which tool radius compensation has been selected (via G16, G17, G18, or G19), functions G41 or G42 must be used to activate it.

G41 Compensation of tool radius compensation to the left.
G42 Compensation of tool radius compensation to the right.

In the same block (or a previous one) in which G41 or G42 is programmed, functions **T, D,** or only **T** must be programmed so that the tool offset value to be applied can be selected from the tool offset table. If no tool offset is selected, the CNC takes D0 with X0 Z0 F0 R0 I0 K0.

When the new selected tool has an M06 associated to it and this M06, in turn, has a subroutine associated to it; the CNC will activate the tool radius compensation at the first movement block of that subroutine.

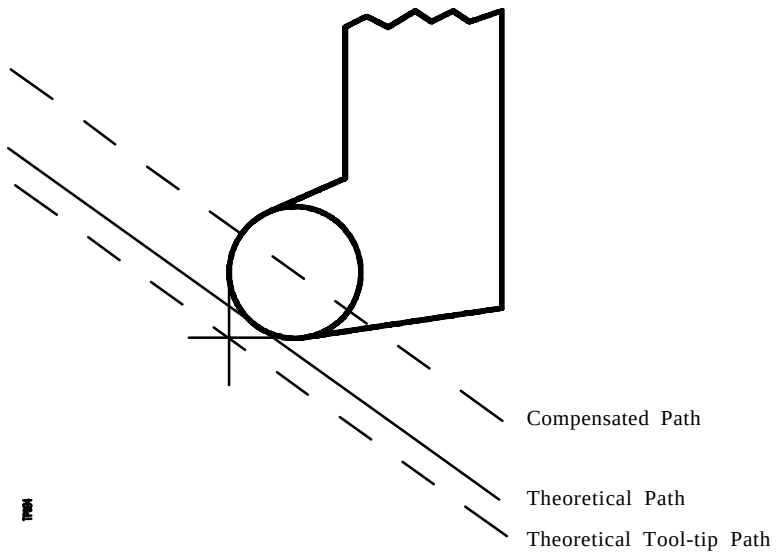
If that subroutine has a G53 programmed in a block (position values referred to Machine Reference Zero, home), the CNC will cancel any tool radius compensation (G41 or G42) selected previously.

The selection of tool radius compensation (G41 or G42) can only be made when functions G00 or G01 are active (straight-line movements).

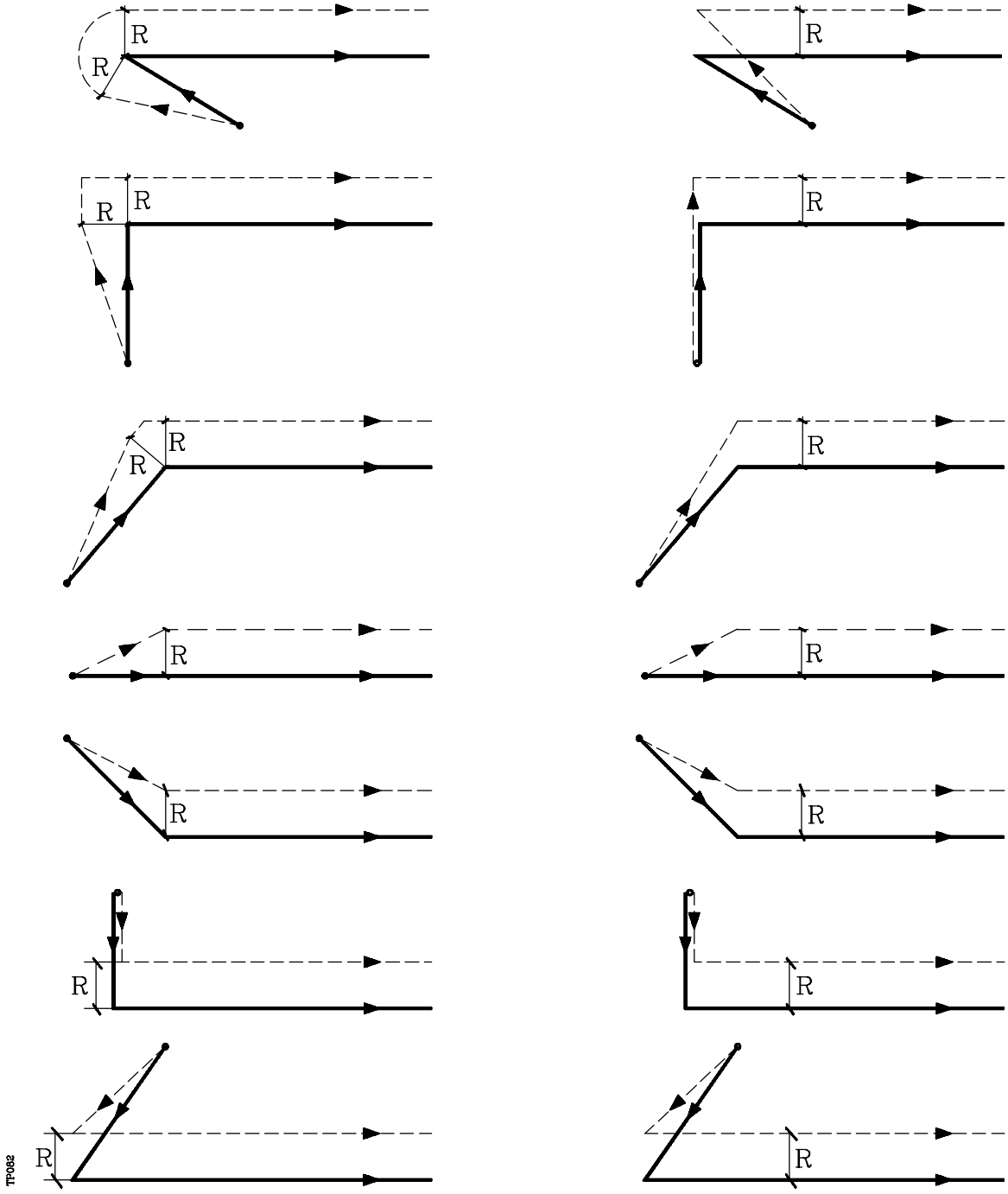
If the compensation is selected while G02 or G03 are active, the CNC will display the corresponding error message.

The following pages show different cases of starting tool radius compensation, in which the programmed path is represented by a solid line and the compensated path with a dotted line.

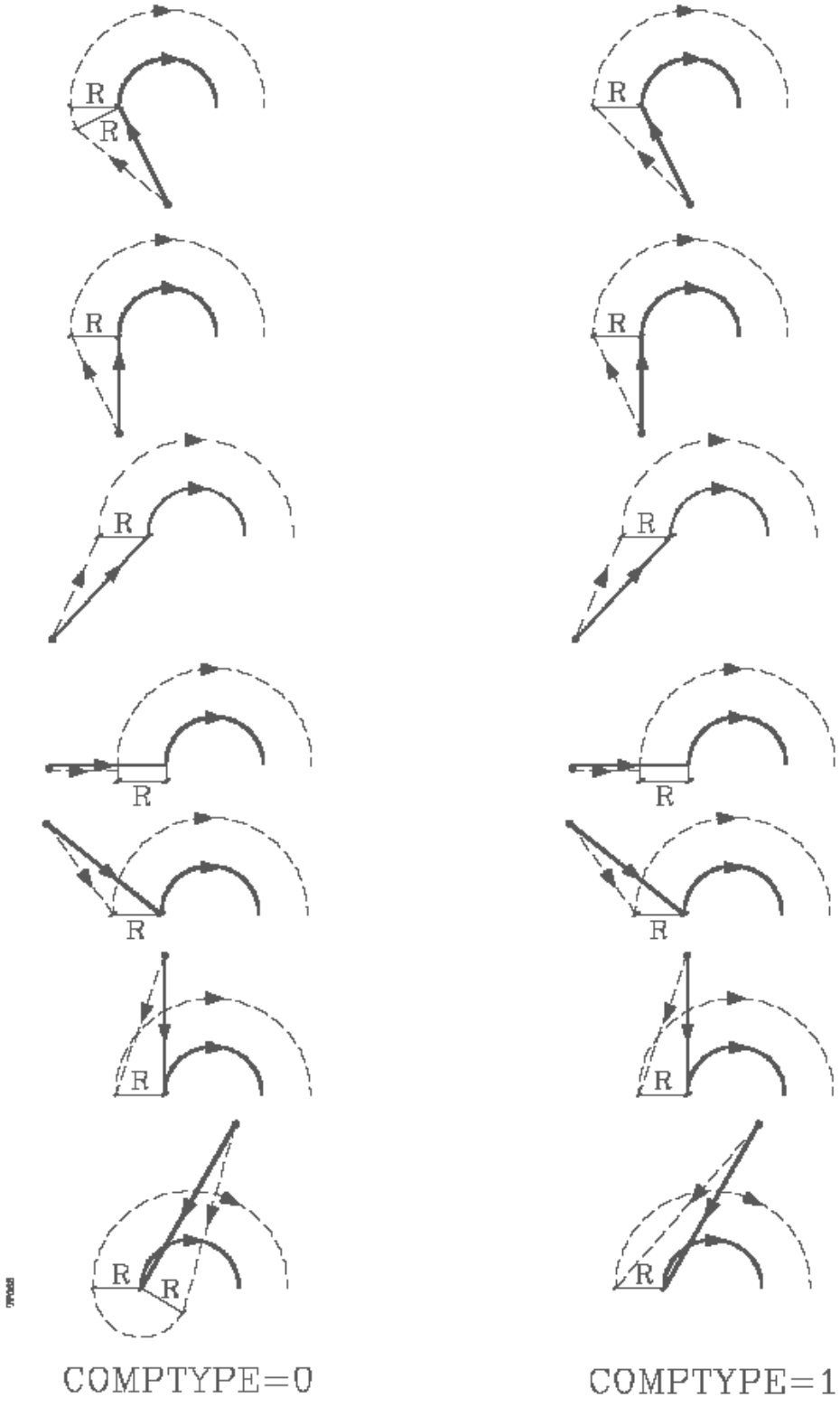
It must be borne in mind that the compensated path corresponds to the tool-tip's center and that the graphic representation shows the theoretical path of the tool tip.



STRAIGHT-STRAIGHT path



STRAIGHT-CURVED path

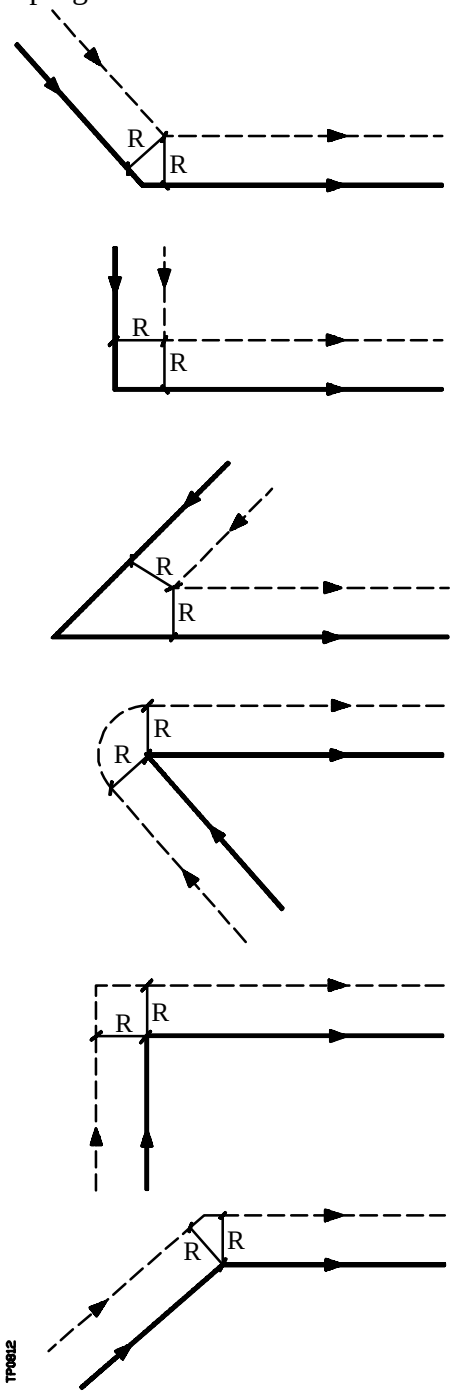


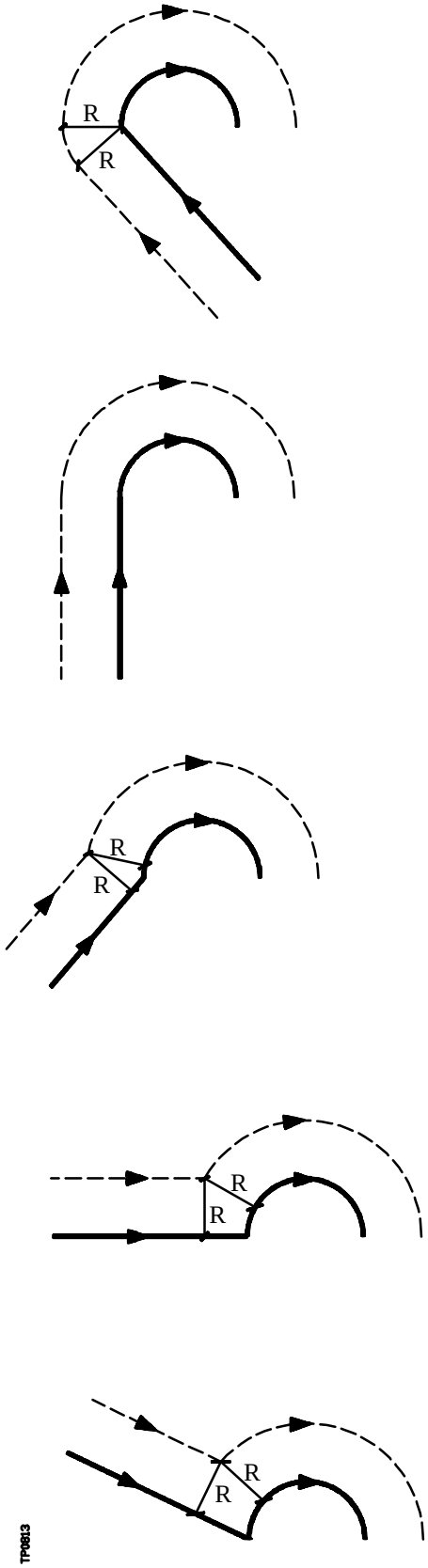
8.3 TOOL RADIUS COMPENSATION SECTIONS

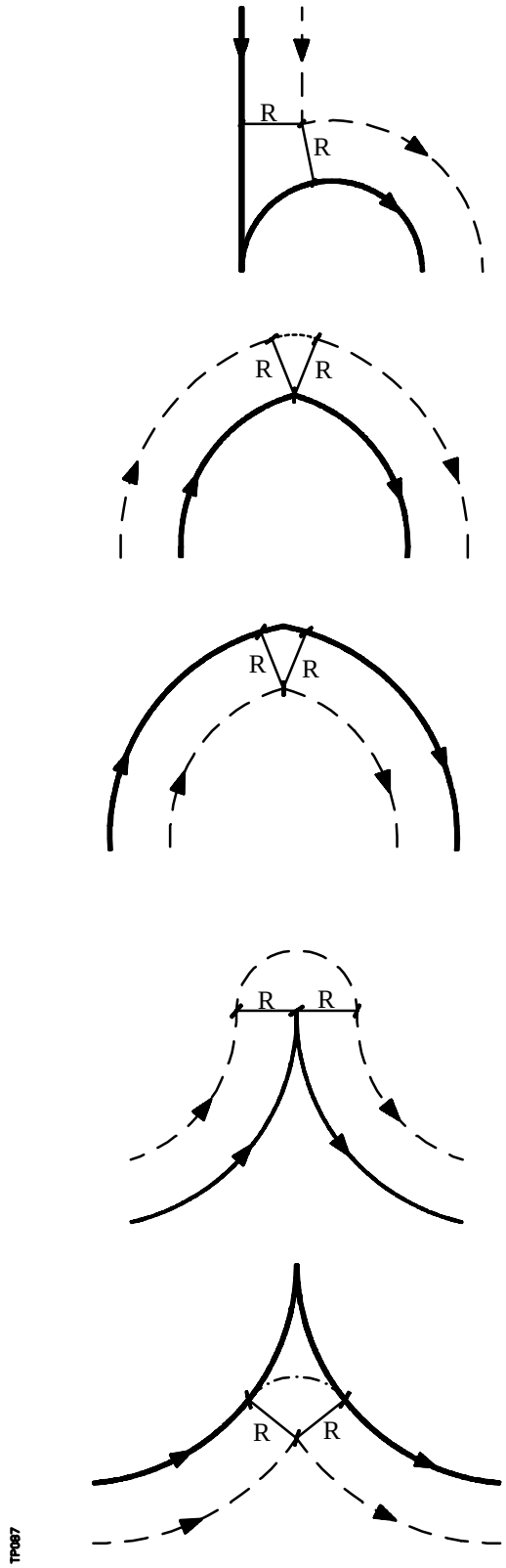
The diagrams (below) show the different paths followed by a tool controlled by a programmed CNC with tool radius compensation.

The programmed path is represented by a solid line and the compensated path by a dotted line. The CNC reads up to 20 blocks ahead of the one it is executing, with the aim of calculating in advance the path to be followed.

When the CNC works with compensation it needs to know the next programmed movement to calculate the path to be followed. For this reason, no more than 17 consecutive blocks can be programmed without movement.




R
R
R
R
R
R
TP0813


R
R
R
R
R
R
R
R
R
R
TP087

8.4 CANCELLING TOOL RADIUS COMPENSATION

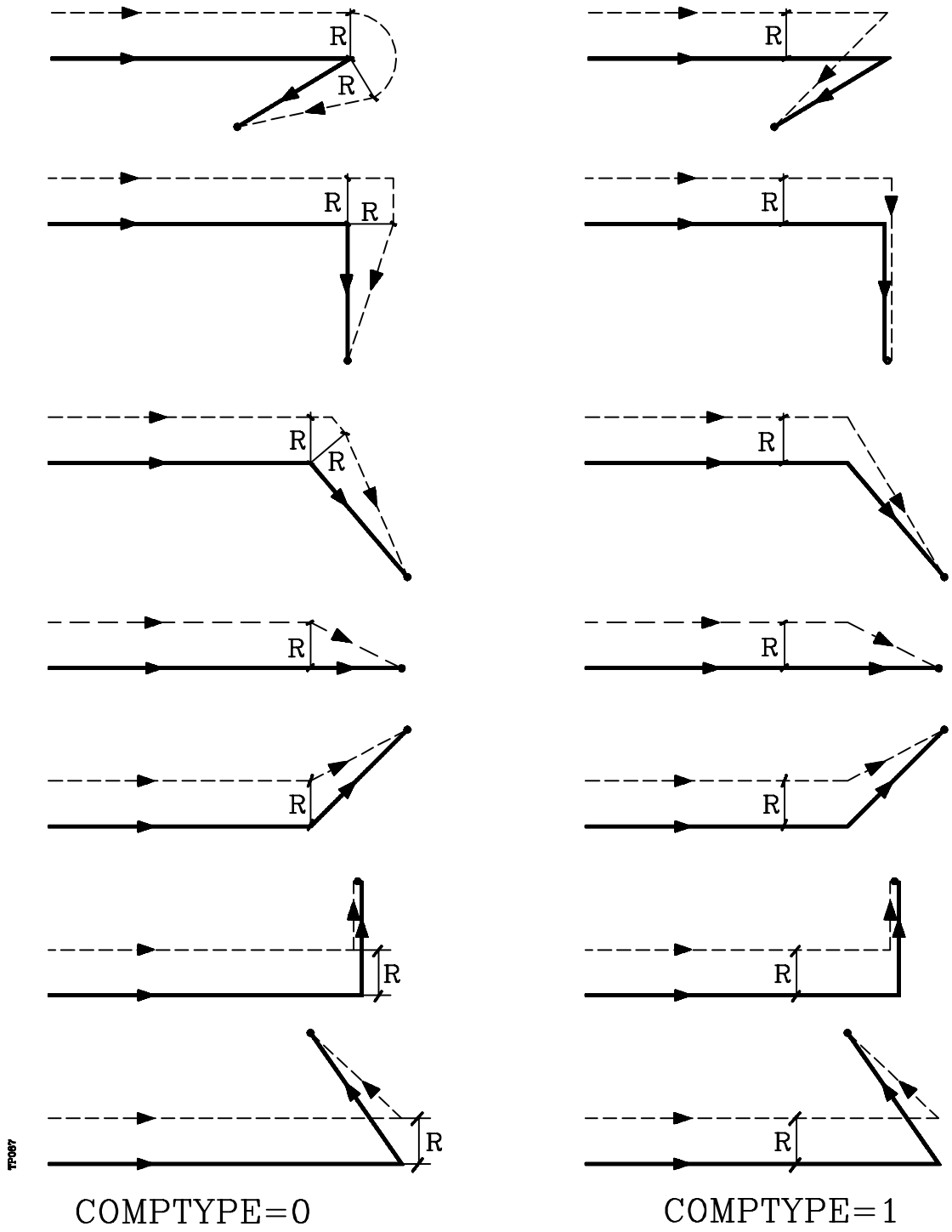
Tool radius compensation is cancelled by using function **G40**.

It should be remembered that cancelling radius compensation (G40) can only be done in a block in which a straight-line movement is programmed (G00 or G01).

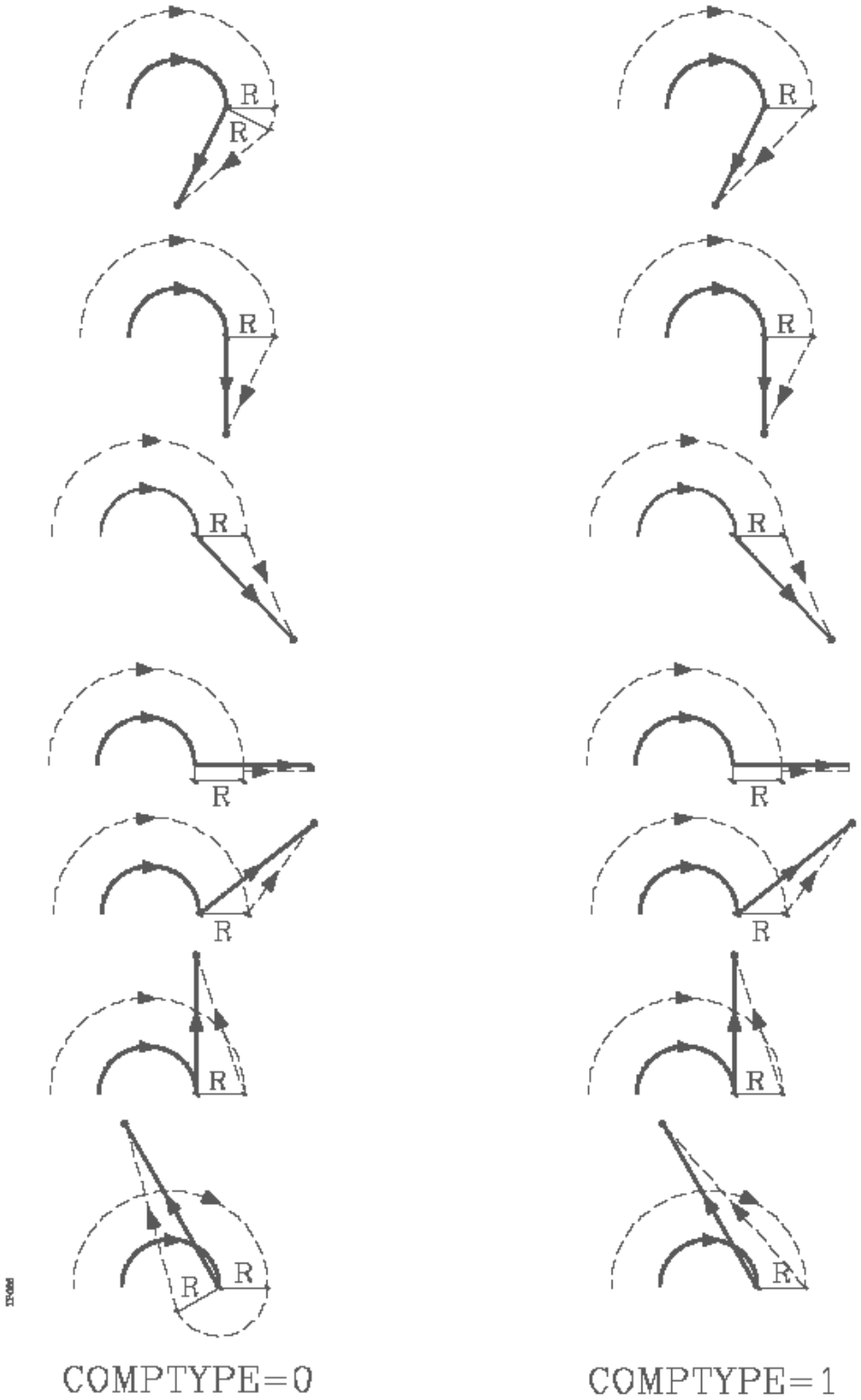
If G40 is programmed while functions G02 or G03 are active, the CNC displays the corresponding error message.

The following pages show different cases of cancelling tool radius compensation, in which the programmed path is represented by a solid line and the compensated path with a dotted line.

STRAIGHT-STRAIGHT path



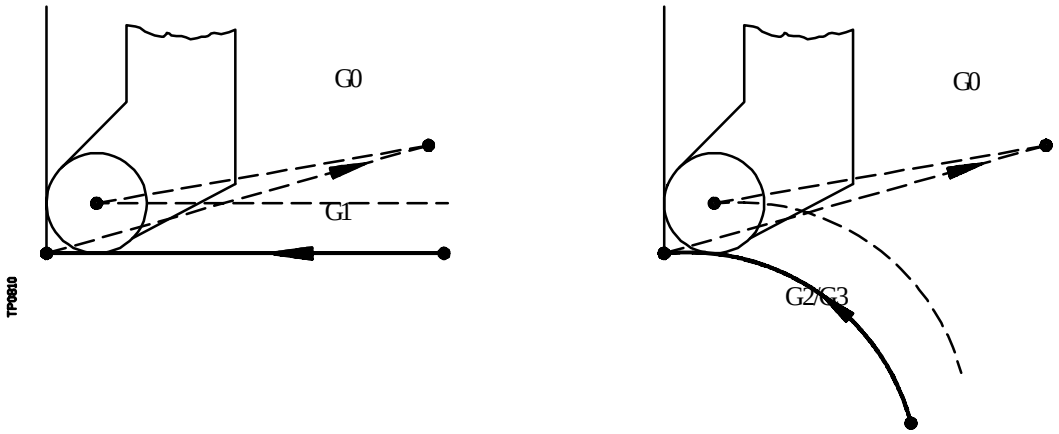
CURVED-STRAIGHT path



8.5 TEMPORARY TOOL RADIUS CANCELLATION WITH G00

Whenever the CNC detects a transition from G01, G02, G03 or G33 to G00, it temporarily cancels tool radius compensation in such a way that the tool will not exceed the line perpendicular to the G01, G02, G03 or G33 path at its end-point.

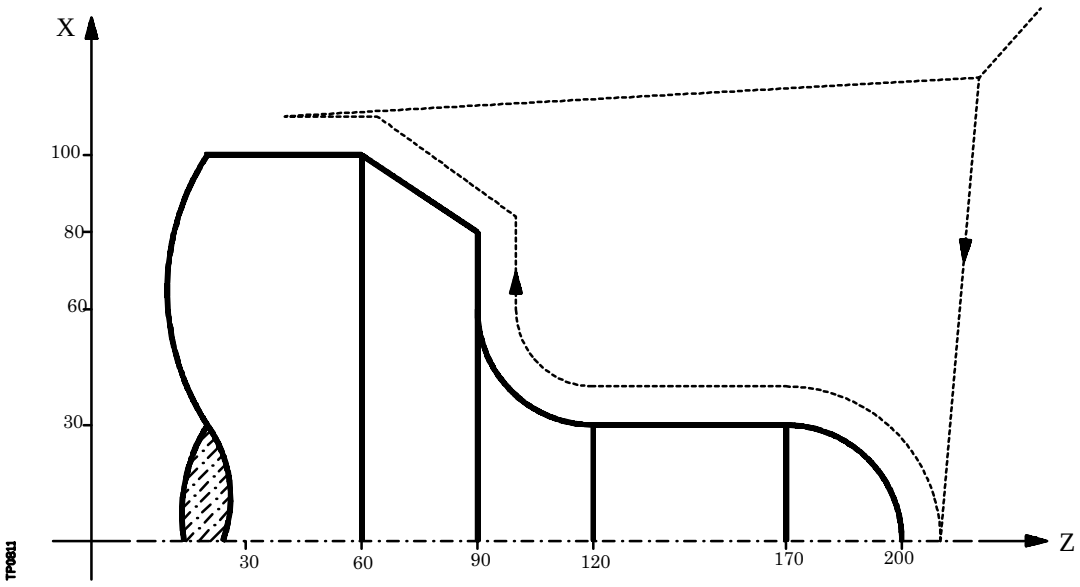
Whenever the CNC detects a transition from G00 to G01, G02, G03 or G33, the new



block receives the treatment corresponding to the first compensated point, resuming tool radius compensation as usual.

Special case: If the CNC does not have enough information for compensation, but it is a G00 move; this move will be carried out without tool radius compensation.

Machining example in diameter with tool radius compensation.



The programmed path is shown with a solid line and the compensated path with a dotted line.

Tool Number : T1
Tool Offset Number : D1

```
G90  G00  X240  Z220  T1   D1     ;  Tool, Offset
G42  G01  X0    Z200                ;  Activate compensation
     G03  X60   Z170  I0   K-30
     G01        Z120
     G02  X120  Z90   I30  K0
     G01  X160
          X200  Z30
G40  G00  X240  Z220                ;  Cancel compensation
M30
```

8.6 TOOL COMPENSATION IN ANY PLANE

With general machine parameter "PLACOM", it is possible to work with tool compensation either in all planes or only in the ZX plane (see chapter on machine parameters in the installation manual).

When setting "PLACOM=1", to work with tool compensation in all planes, the CNC interprets the tool table as follows:

	ZX plane	WX plane	AB plane
Parameters Z and K, with the abscissa axis	Z axis	W axis	A axis
Parameters X and I, with the ordinate axis	X axis	X axis	B axis

9. *CANNED CYCLES*

The CNC offers the following machining canned cycles:

- **G66** Pattern repeat.
- **G68** Stock removal along the X axis.
- **G69** Stock removal along the Z axis.
- **G81** Turning cycle with straight sections.
- **G82** Facing cycle with straight sections.
- **G83** Drilling cycle.
- **G84** Turning cycle with circular sections.
- **G85** Facing cycle with circular sections.
- **G86** Longitudinal threadcutting.
- **G87** Face threadcutting.
- **G88** Grooving cycle along X axis.
- **G89** Grooving cycle along Z axis.

Machining canned cycles with a live tool:

- **G60** Drilling / tapping canned cycle on the face of the part
- **G61** Drilling / tapping canned cycle along the surface of the part
- **G62** Slot milling canned cycle along the surface of the part.
- **G63** Slot milling canned cycle on the face of the part.

A canned cycle is defined by the G function indicating the canned cycle and its corresponding parameters.

A canned cycle may be defined anywhere in the program, that is, in the main program as well as in a subroutine.

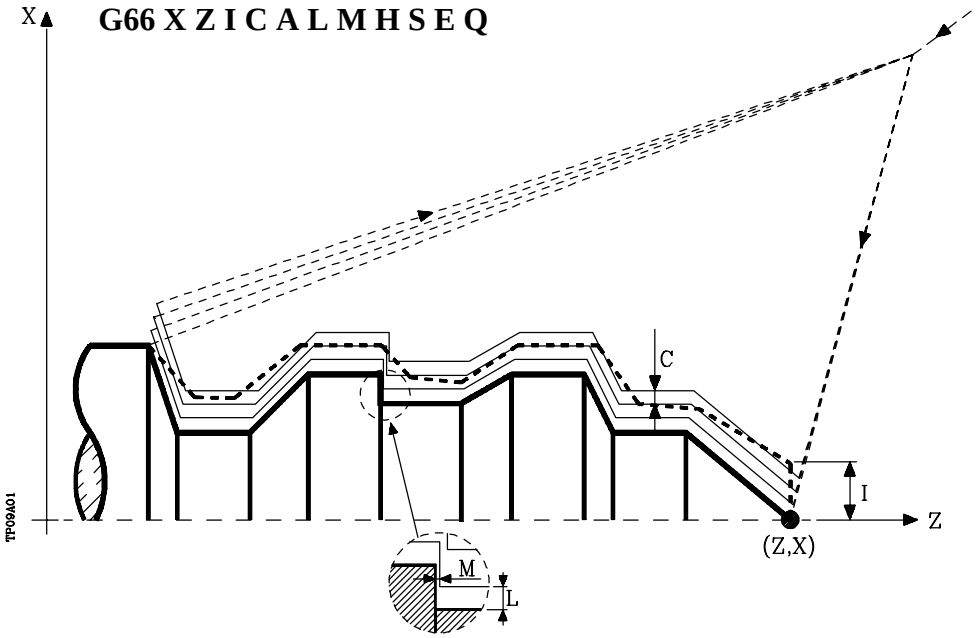
When working in a plane other than the ZX, for example: G16 WX, the CNC interprets the canned cycle parameters as follows:

	ZX plane	WX plane	AB plane
Parameter Z and all related to it, with the abscissa axis	Z axis	W axis	A axis
Parameter X and all related to it, with the ordinate axis	X axis	X axis	B axis

9.1 G66. PATTERN REPEAT CANNED CYCLE

This cycle machines the programmed profile maintaining the specified pass between the successive machining passes.

The basic structure of the block is:



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

I5.5 Defines the residual stock, i.e. the amount to be removed from the original part. It is given in radius and it will be taken as X residual or Z residual depending on the value given to parameter "A".

If "H" is other than zero and "I" is not greater than the finishing stock (L or M), The CNC will only run the finishing pass.

C5.5 Defines the machining pass. It is given in radius and it will be taken as pass along X or along Z depending on the value given to parameter "A".

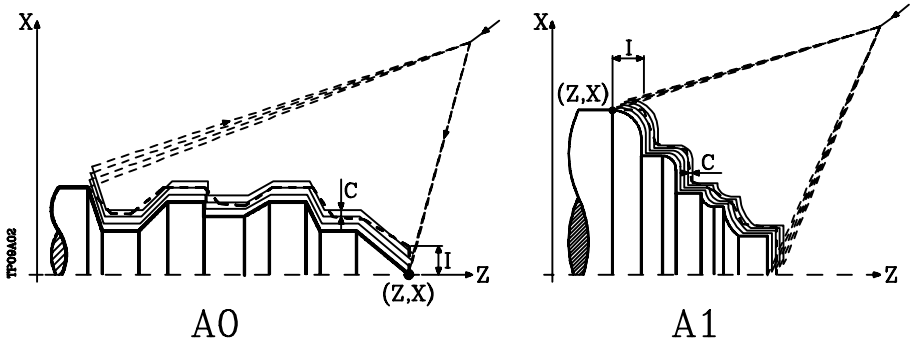
All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

If programmed with a 0 value, the CNC will display the corresponding error message.

A1 Defines the main machining axis.

When programming "A0", the Z axis will be the main axis, "I" will be taken as X residual value and "C" as the pass along X.

When programming "A1", the X axis will be the main axis, "I" will be taken as Z residual value and "C" as the pass along Z.



If parameter A is not programmed, the "I" and "C" values depend on the tool dimensions.

If the tool length along X is greater than along Z, the "I" value is taken as residual along X and the "C" value as the pass along X.

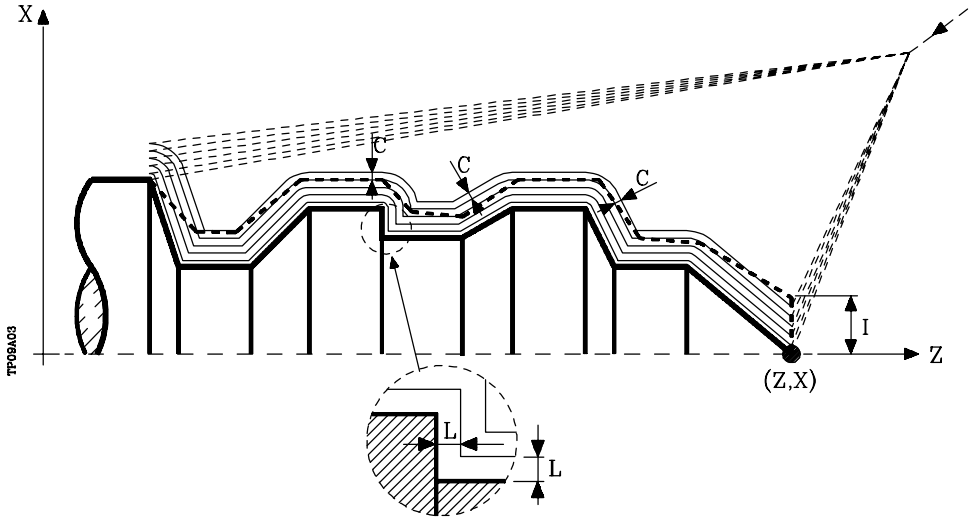
If the tool length along Z is greater than along X, the "I" value is taken as residual along Z and the "C" value as the pass along Z.

L5.5 Defines the finishing pass along X. It is given in radius and, if not programmed, a value of 0 will be assumed.

M±5.5 Defines the finishing pass along Z.

If either "L" or "M" is negative, the finishing pass will be carried out in round corner (G05). When they are both positive, the finishing pass will be performed in square corner (G07).

If "M" is not programmed, the X and Z residual value will be the one indicated by parameter "L" and the roughing passes will be the same maintaining the "C" distance between two consecutive passes.



H5.5 Defines the feedrate for the finishing pass.

If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

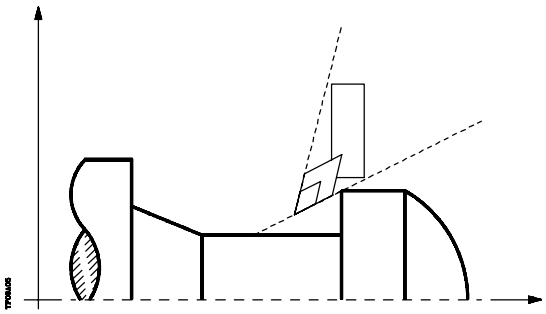
S4 Defines the label number of the first block describing the geometry of the profile.

E4 Defines the label number of the last block describing the geometry of the profile.

Q6 Defines the number of the program containing the geometry description of the profile. This parameter is optional and if not set, the CNC interprets that the profile is defined within the same program that contains the cycle.

Basic operation:

* In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.



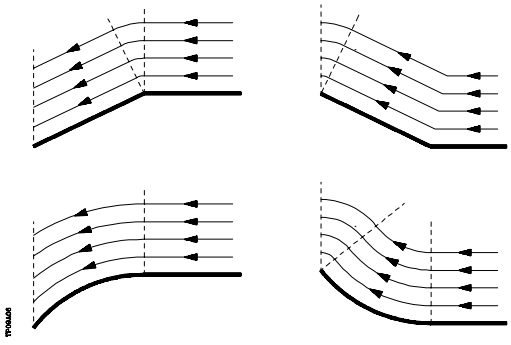
The operator will be able to stop the execution and select the proper tool.

If he/she doesn't do that, a new profile is calculated in the areas that are not accessible for the selected tool and the CNC machines as far as it is possible. The message will remain displayed during the whole machining process.

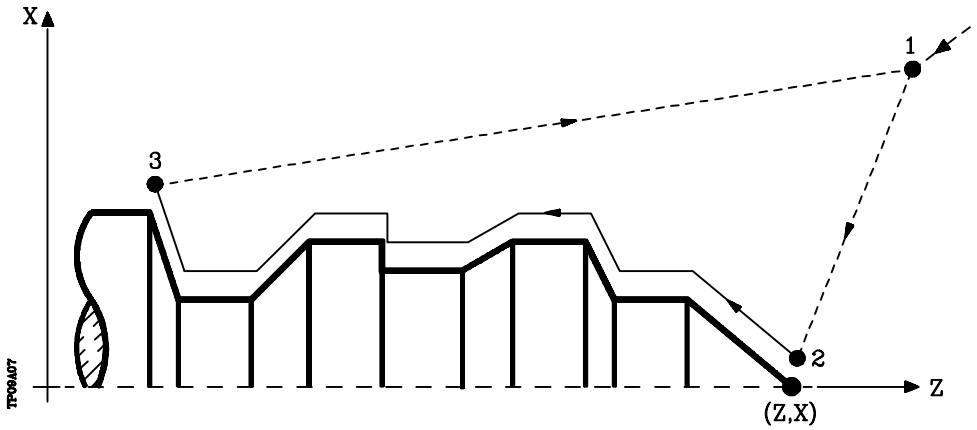
* Once the profile to be executed has been calculated, the CNC will calculate all the passes required to eliminate the programmed residual stock (I).

The whole machining operation will be performed in whatever G05 or G07 that was active when calling the cycle.

When not programming parameter "M", the passes will be the same maintaining the "C" distance between 2 consecutive passes. Also, if the last section of the profile is an arc or an incline plane, the CNC will calculate the different passes without exceeding the maximum programmed position.

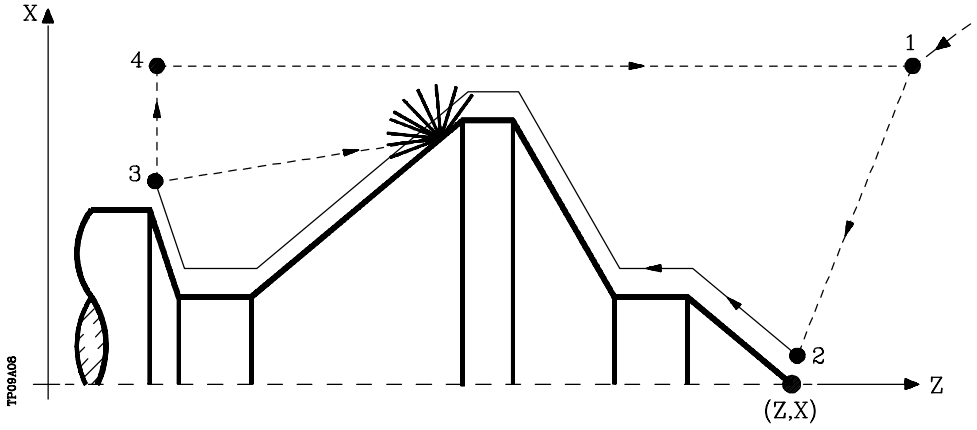


* Each pass is performed in the following way:



- The approach move “1-2” is done in rapid (G00).
- The “2-3” move is done at programmed feedrate (F).
- The withdrawal “3-1” is done in rapid (G00).

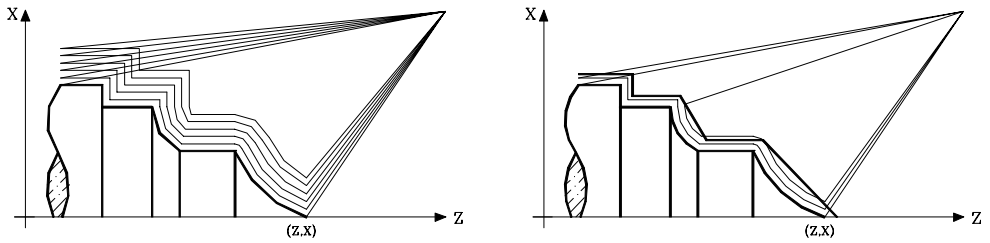
  If the possibility of collision exists, this move will be carried out in two stages and in G00: “3-4” and “4-1” as shown in the figure on the next page.



* The canned cycle will always end at the point from where it was called.

Optimizing the machining process

If only the desired profile is defined, the CNC assumes that the rough part is cylindrical and it machines it as shown in the drawing on the left.



When the profile of the rough part is known, it is recommended to define both profiles: the rough part and the desired final profile. The machining is faster since only the stock between both profiles is removed.

Profile programming syntax

When defining the profile, there is no need to program the initial point since it is already indicated by cycle defining parameters X and Z.

When defining 2 profiles, the final profile must be defined first and then that of the rough part.

The first and last profile defining blocks must have a label number. These labels will indicate the beginning and end of the profile geometry definition for this canned cycle.

The profile programming syntax must follow these rules:

1.- It may be programmed in absolute or incremental coordinates and it may consist of simple geometrical elements such as straight and curved lines, roundings and chamfers which will be programmed following their own syntax rules.

2.- Function G00 indicates that the profile definition is done and this block is the beginning of the definition of the rough part profile.

Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from displaying the corresponding error message.

3.- The profile description must not contain: mirror images, scaling factor changes, pattern rotation or zero offsets.

4.- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.

5.- It must not contain other canned cycles.

Only the following G functions may be programmed in the profile definition:

G00 Positioning in rapid
G01 Linear interpolation
G02 Clockwise circular interpolation
G03 Counter-clockwise circular interpolation
G06 Circle center in absolute coordinates
G08 Arc tangent to previous path
G09 Arc defined by three points
G36 Automatic radius blend
G39 Chamfer
G53 Programming with respect to machine zero
G70 Programming in inches
G71 Programming in millimeters
G90 Absolute programming
G91 Incremental programming
G93 Polar origin preset

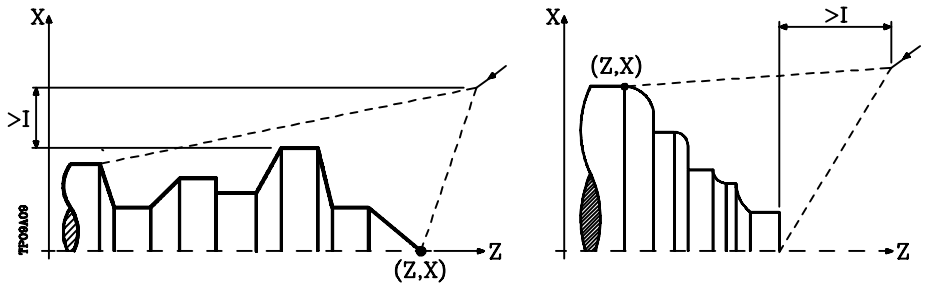
It is also possible to program the following functions although they will be ignored by the canned cycle:

G05 Round corner
G07 Square corner
G50 Controlled corner rounding
The F, S, T, D and M functions.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the canned cycle.

The canned cycle calling position must be off the part to be machined and at a distance greater than that defined as residual stock (I) of the outmost profile of the part.



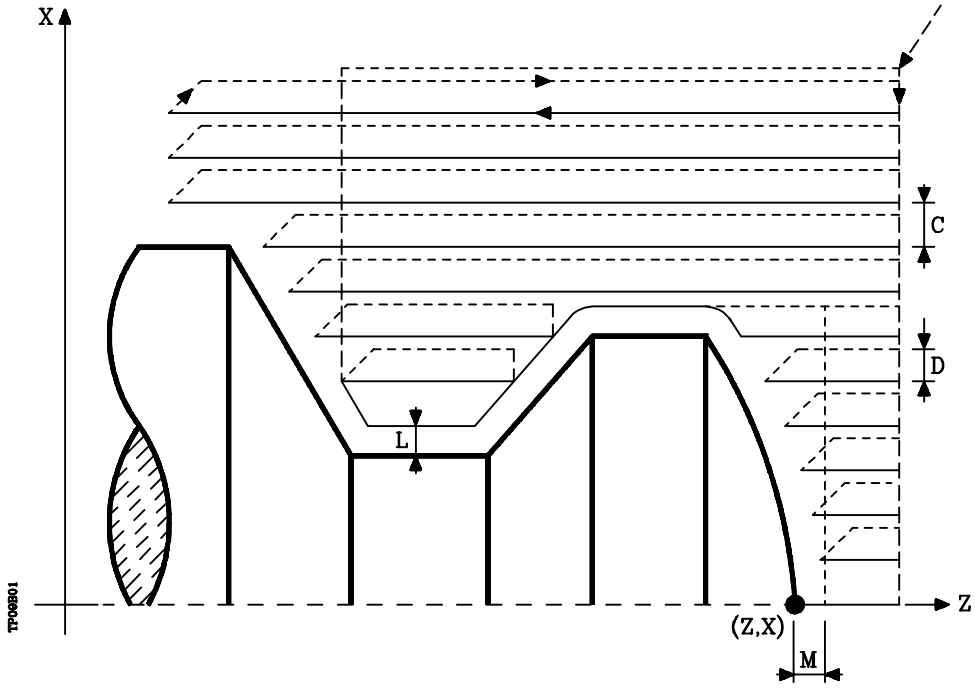
If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.

Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or the finishing operation (H). The CNC will assume functions G00, G40 and G90.

9.2 G68. STOCK REMOVAL CYCLE ALONG X AXIS

This cycle machines the programmed profile maintaining the specified pass between the successive machining passes.

The basic structure of the block is: **G68 X Z C D L M K F H S E Q**



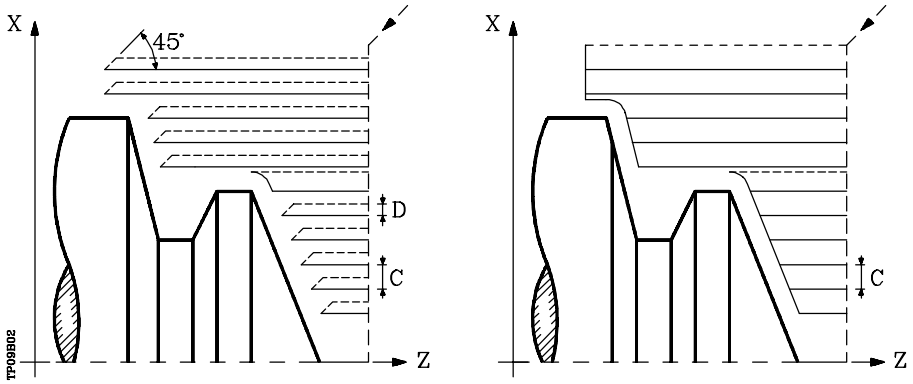
X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

C5.5 Defines the machining pass. It is given by a positive radius value. If programmed with a 0 value, the CNC will display the corresponding error message.

All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

D 5.5 Defines the safety distance to withdraw the tool after each pass.



When programming "D" with a value other than "0", the cutter withdraws at a 45º angle up to the safety distance (left drawing).

When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing).

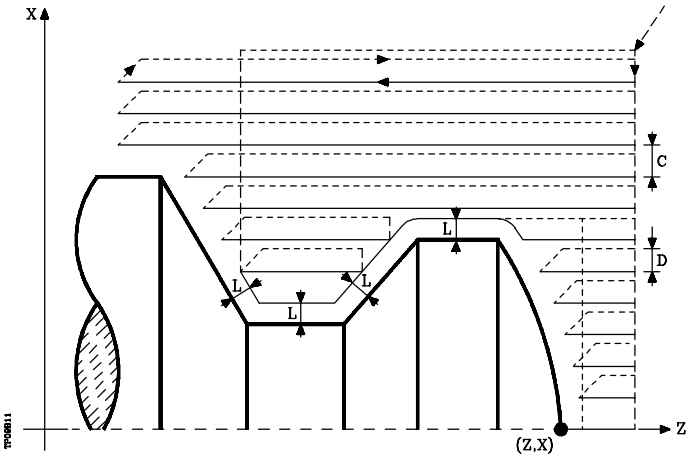
It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.

L±5.5 Defines the finishing pass along X. It is given in radius and, if not programmed, a value of 0 will be assumed.

M±5.5 Defines the finishing pass along Z.

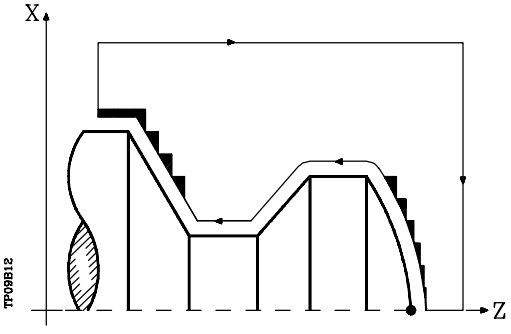
If either "L" or "M" is negative, the finishing pass will be carried out in round corner (G05). When they are both positive, the finishing pass will be performed in square corner (G07).

If "M" is not programmed, the X and Z residual value will be the one indicated by parameter "L" and the roughing passes will be the same maintaining the "C" distance between two consecutive passes.



K5.5 Defines the penetration feedrate for the "valleys". If not programmed or programmed with a value of "0", it assumes the machining feedrate that was programmed before calling the cycle.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



H5.5 Defines the feedrate for the finishing pass. If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

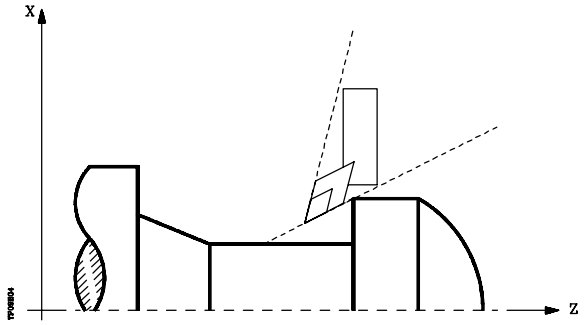
S4 Defines the label number of the first block describing the geometry of the profile.

E4 Defines the label number of the last block describing the geometry of the profile.

Q6 Defines the number of the program containing the geometry description of the profile. This parameter is optional and if not set, the CNC interprets that the profile is defined within the same program that contains the cycle.

Basic operation:

* In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.

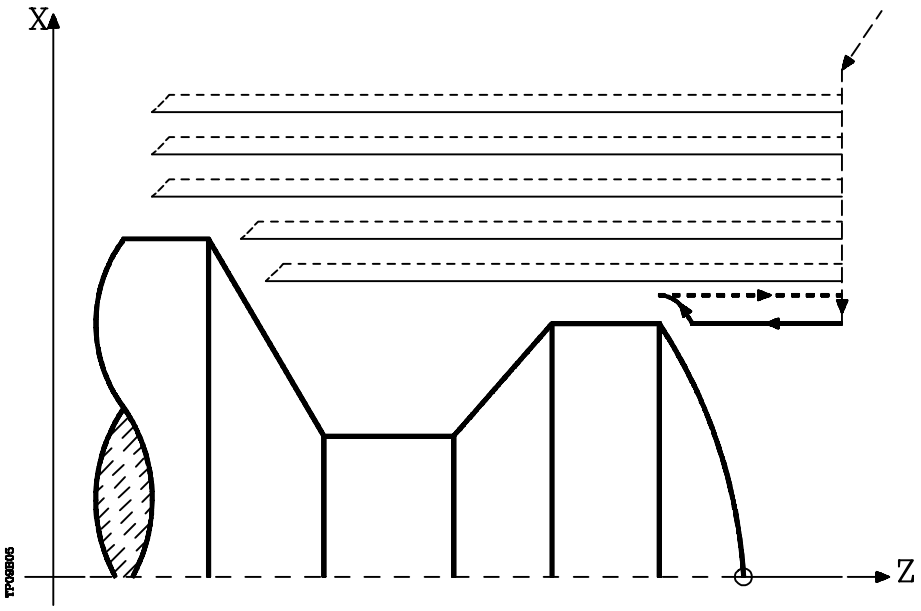


The operator will be able to stop the execution and select the proper tool.

If he/she doesn't do that, a new profile is calculated in the areas that are not accessible for the selected tool and the CNC machines as far as it is possible. The message will remain displayed during the whole machining process.

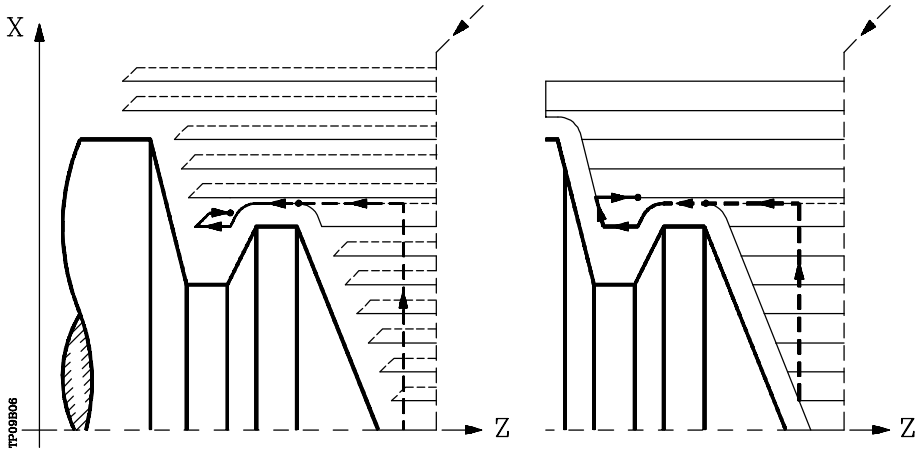
The whole machining operation will be performed in whatever G05 or G07 that was active when calling the cycle and it will be carried out with the same pass throughout the whole machining operation.

* If when making a roughing pass a channel is detected, the CNC will continue the operation ignoring the channel.

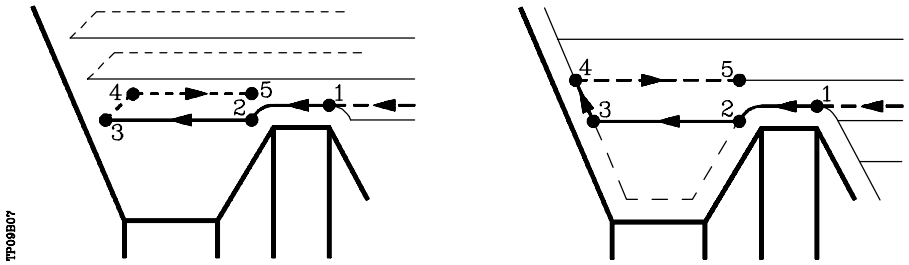


* The profile may have an unlimited number of channels.

* Once the residual profile is machined, the detected channels will be machined.



To do so, it will return in G00 to the point where the profile machining was interrupted (1).

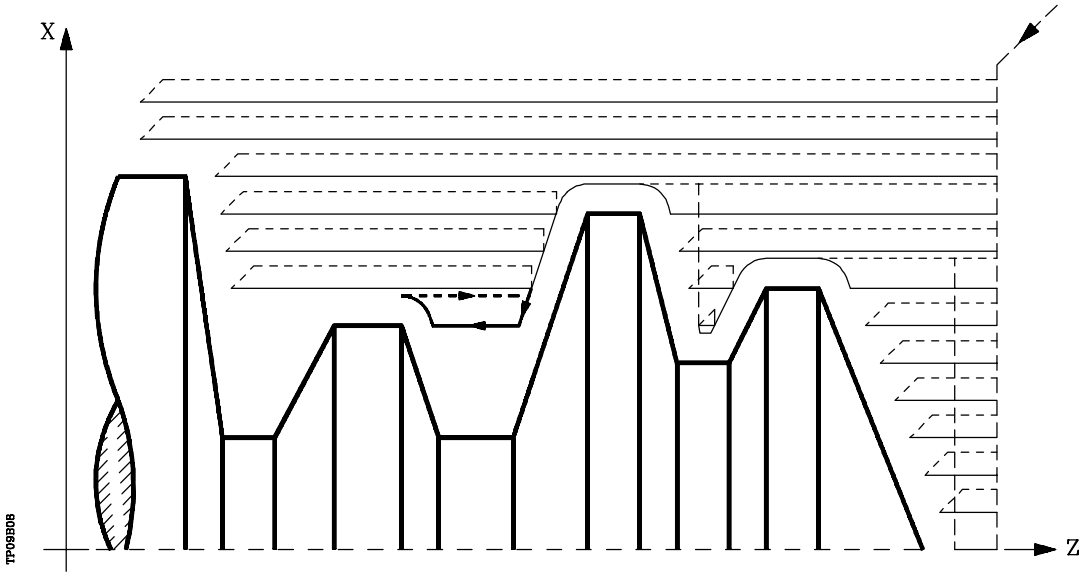


From this point on, it will follow the programmed contour maintaining the finishing stock until the selected pass “C” is reached. Section 1-2.

The new roughing pass is performed as follows:

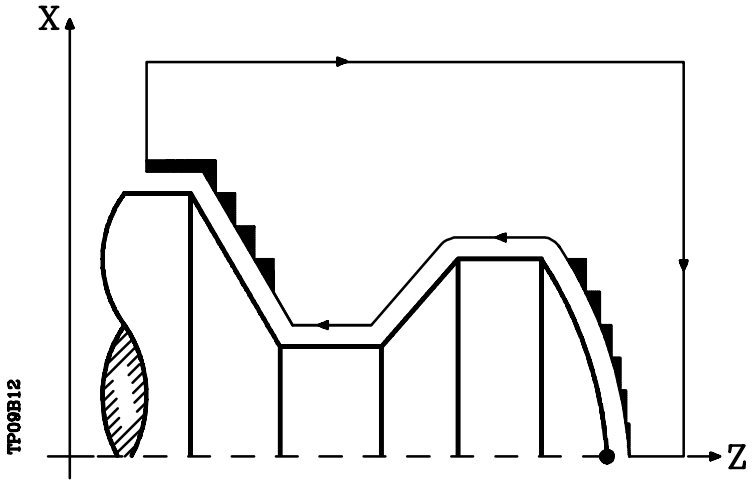
- Move “2-3” in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- Withdrawal “4-5” in rapid (G00).

* If when machining a channel, other channels inside it are detected, the same procedure explained above will be followed.



* If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F".

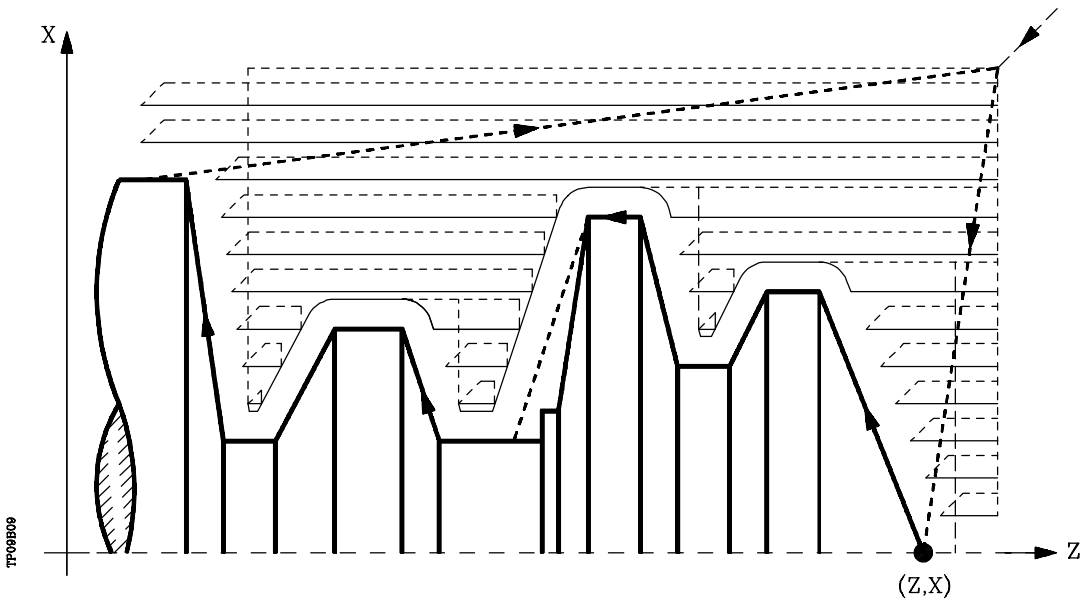
  This final roughing pass will eliminate the ridges that were left in the roughing stage.



* Once the profile roughing operation has ended, the tool will return to the point from where the cycle was called.

* If a finishing pass has been defined, it will be carried out with tool radius compensation and at the indicated feedrate “H”.

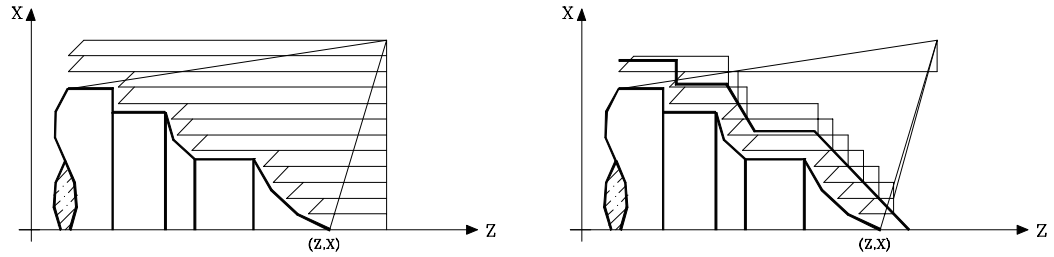
  This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.



Once the finishing pass has ended, the tool will return to the point from where the cycle was called.

Optimizing the machining process

If only the desired profile is defined, the CNC assumes that the rough part is cylindrical and it machines it as shown in the drawing on the left.



When the profile of the rough part is known, it is recommended to define both profiles: the rough part and the desired final profile. The machining is faster since only the stock between both profiles is removed.

Profile programming syntax

When defining the profile, there is no need to program the initial point since it is already indicated by cycle defining parameters X and Z.

When defining 2 profiles, the final profile must be defined first and then that of the rough part.

The first and last profile defining blocks must have a label number. These labels will indicate the beginning and end of the profile geometry definition for this canned cycle.

The profile programming syntax must follow these rules:

1.- It may be programmed in absolute or incremental coordinates and it may consist of simple geometrical elements such as straight and curved lines, roundings and chamfers which will be programmed following their own syntax rules.

2.- Function G00 indicates that the profile definition is done and this block is the beginning of the definition of the rough part profile.

Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from displaying the corresponding error message.

3.- The profile description must not contain: mirror images, scaling factor changes, pattern rotation or zero offsets.

4.- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.

5.- It must not contain other canned cycles.

Only the following G functions may be programmed in the profile definition:

G00 Positioning in rapid
G01 Linear interpolation
G02 Clockwise circular interpolation
G03 Counter-clockwise circular interpolation

G06 Circle center in absolute coordinates
G08 Arc tangent to previous path
G09 Arc defined by three points
G36 Automatic radius blend
G39 Chamfer
G53 Programming with respect to machine zero
G70 Programming in inches
G71 Programming in millimeters
G90 Absolute programming
G91 Incremental programming
G93 Polar origin preset

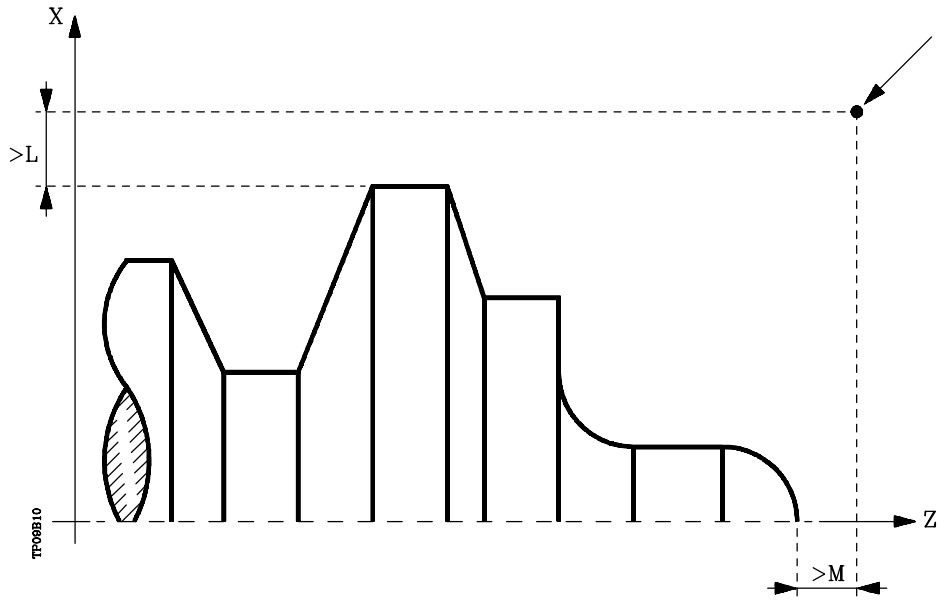
It is also possible to program the following functions although they will be ignored by the canned cycle:

G05 Round corner
G07 Square corner
G50 Controlled corner rounding
The F, S, T, D and M functions.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the canned cycle.

The canned cycle calling position must be off the part to be machined and at a distance greater than that defined as finishing stock (L, M) along the X and Z axes.



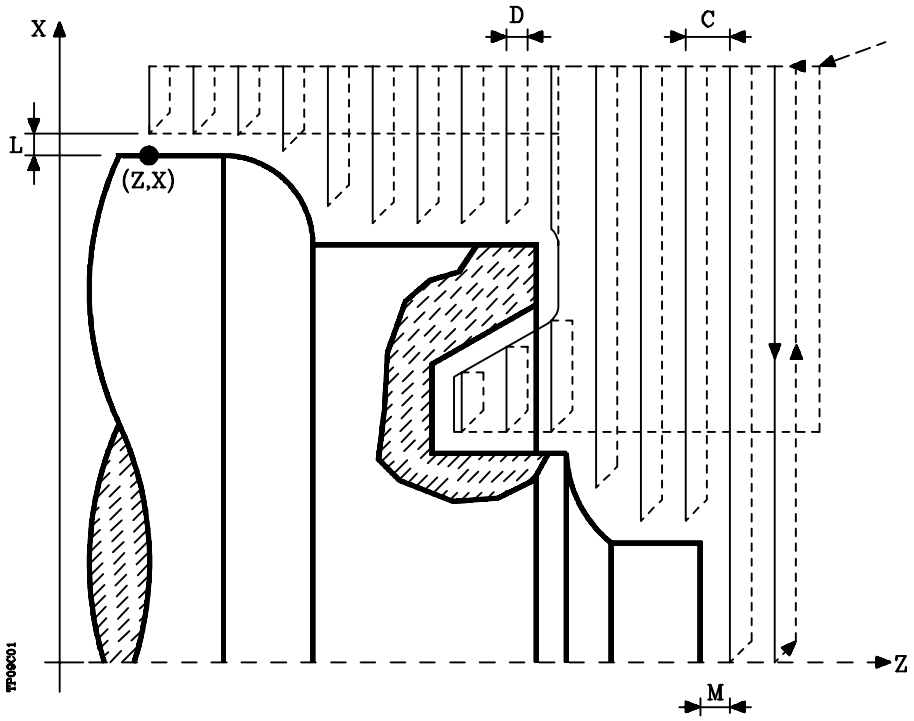
If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.

Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or the finishing operation (H). The CNC will assume functions G00, G40 and G90.

9.3 G69. STOCK REMOVAL CANNED CYCLE ALONG Z AXIS

This cycle machines the programmed profile maintaining the specified pass between the successive machining passes.

The basic structure of the block is: **G68 X Z C D L M K F H S E Q**



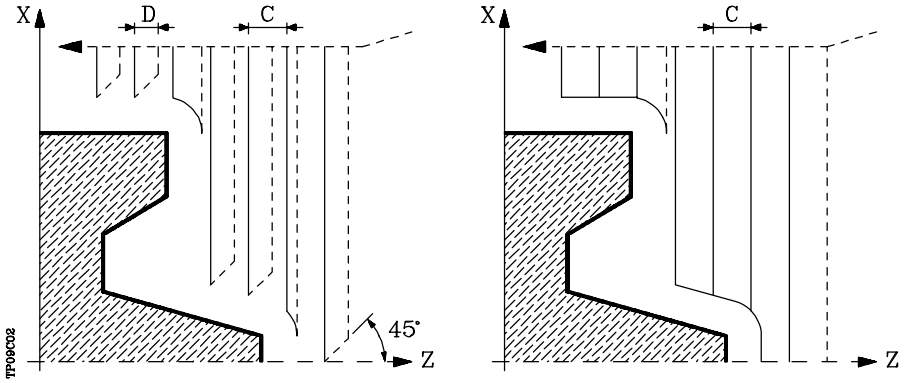
X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

C5.5 Defines the machining pass. It is given by a positive radius value. If programmed with a 0 value, the CNC will display the corresponding error message.

All the machining passes are performed with this value, except the last one which will eliminate the remaining stock.

D 5.5 Defines the safety distance to withdraw the tool after each pass.



When programming "D" with a value other than "0", the cutter withdraws at a 45º angle up to the safety distance (left drawing).
When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing).

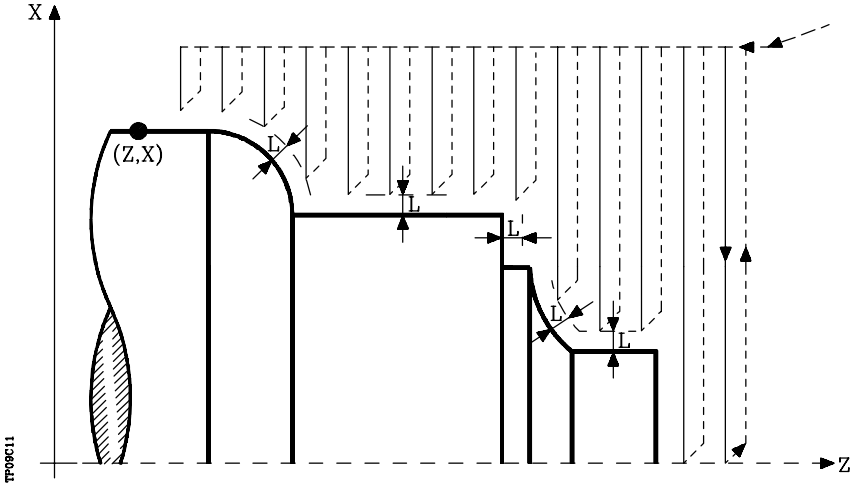
It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.

L±5.5 Defines the finishing pass along X. It is given in radius and, if not programmed, a value of 0 will be assumed.

M±5.5 Defines the finishing pass along Z.

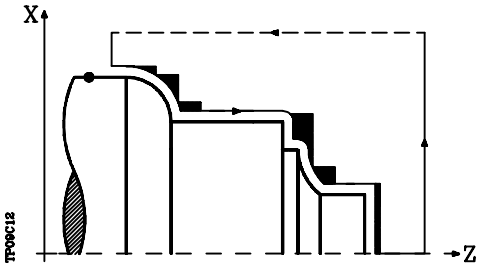
If either "L" or "M" is negative, the finishing pass will be carried out in round corner (G05). When they are both positive, the finishing pass will be performed in square corner (G07).

If "M" is not programmed, the X and Z residual value will be the one indicated by parameter "L" and the roughing passes will be the same maintaining the "C" distance between two consecutive passes.



K5.5 Defines the penetration feedrate for the "valleys". If not programmed or programmed with a value of "0", it assumes the machining feedrate that was programmed before calling the cycle.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



H5.5 Defines the feedrate for the finishing pass. If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

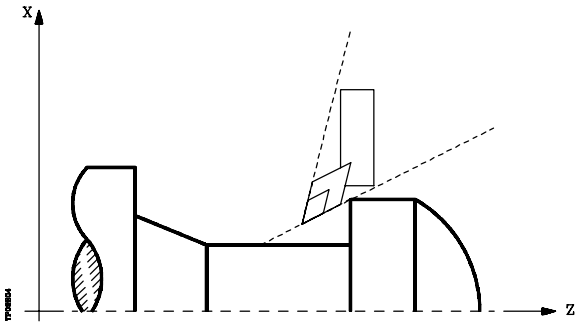
S4 Defines the label number of the first block describing the geometry of the profile.

E4 Defines the label number of the last block describing the geometry of the profile.

Q6 Defines the number of the program containing the geometry description of the profile. This parameter is optional and if not set, the CNC interprets that the profile is defined within the same program that contains the cycle.

Basic operation:

* In those cases when the programmed profile cannot be machined (valleys) with the selected tool, a warning message will be displayed at the beginning of the cycle execution.



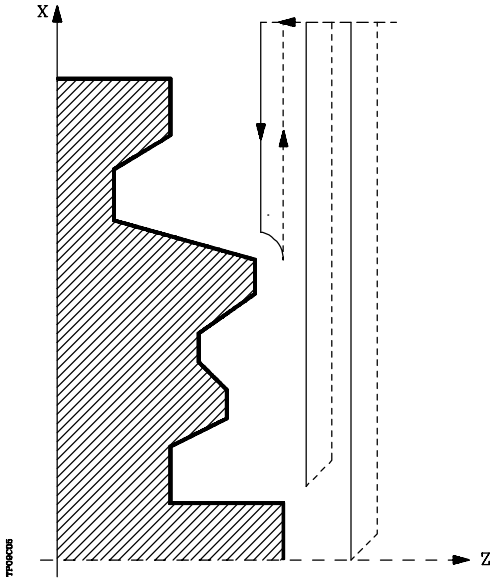
The operator will be able to stop the execution and select the proper tool.

If he/she doesn't do that, a new profile is calculated in the areas that are not accessible for the selected tool and the CNC machines as far as it is possible. The message will remain displayed during the whole machining process.

* Once the necessary roughing passes have been calculated, the CNC will machine the resulting new profile

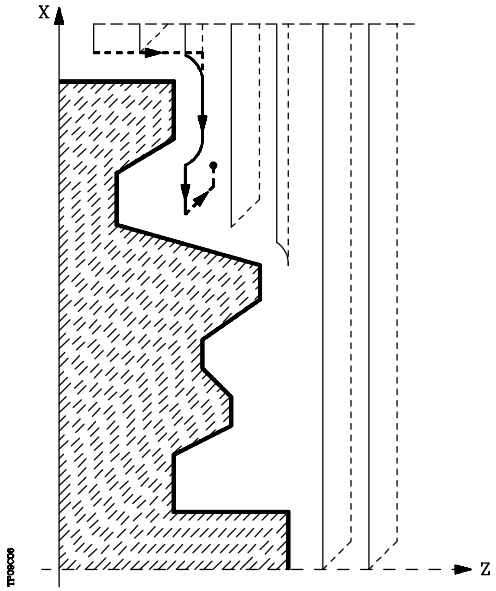
  The whole machining operation will be performed in whatever G05 or G07 that was active when calling the cycle and it will be carried out with the same pass throughout the whole machining operation.

* If when making a roughing pass, a channel is detected, the CNC will continue the operation ignoring the channel.

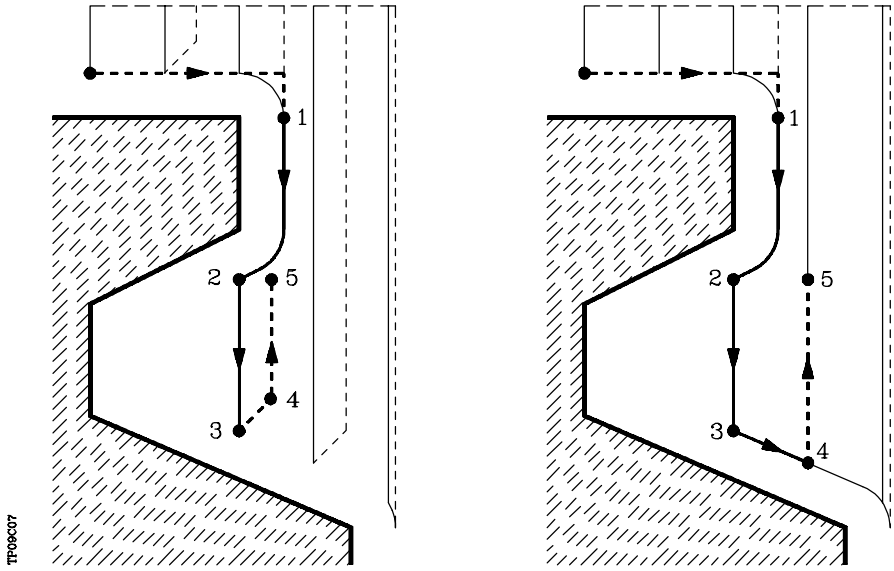


* The profile may have an unlimited number of channels.

* Once the residual profile is machined, the detected channels will be machined.



To do so, it will return in G00 to the point where the profile machining was interrupted (1).

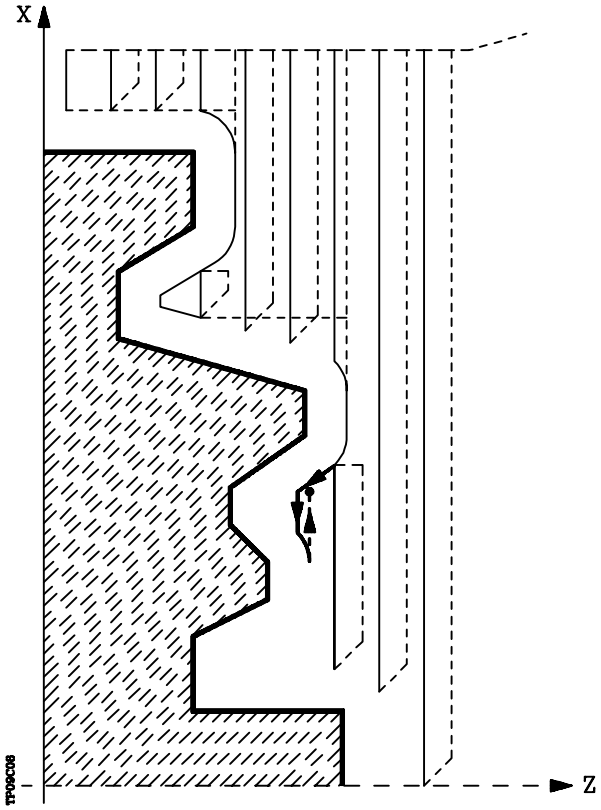


From this point on, it will follow the programmed contour maintaining the finishing stock until the selected pass “C” is reached. Section 1-2.

The new roughing pass is performed in the following manner:

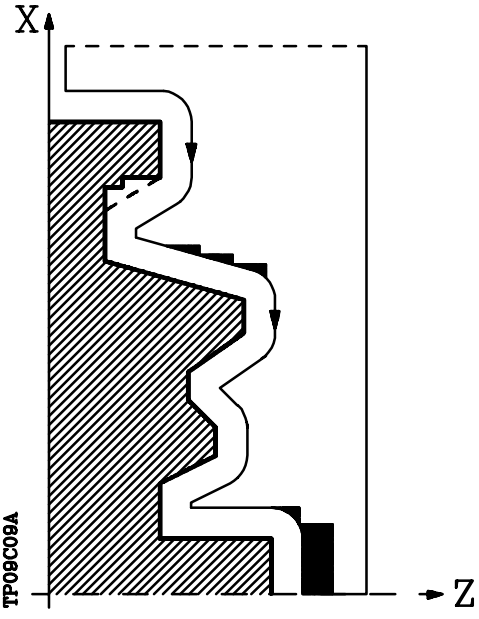
- Move “2-3” in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- Withdrawal “4-5” in rapid (G00).

* If when machining a channel, other channels inside it are detected, the same procedure explained above will be followed.



* If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" stock at the set feedrate "F".

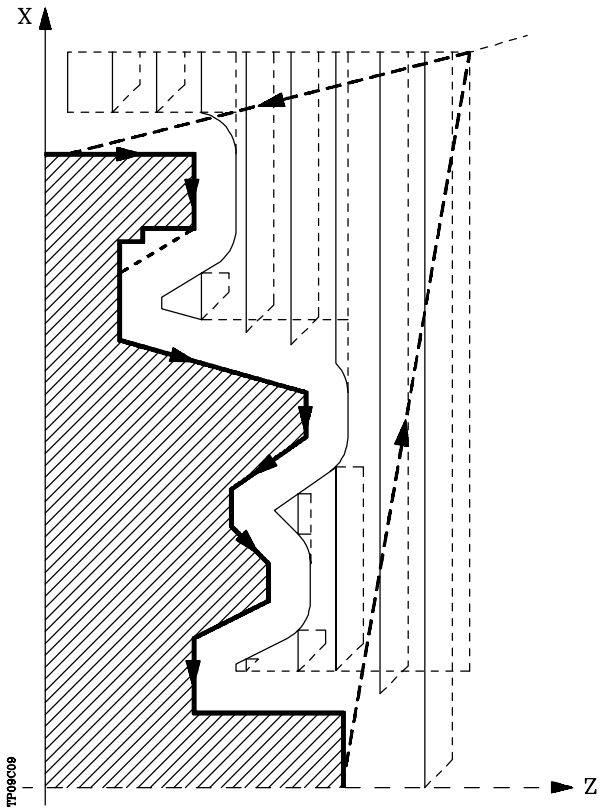
This final roughing pass will eliminate the ridges that were left in the roughing stage.



* Once the profile roughing operation has ended, the tool will return to the point from where the cycle was called.

* If a finishing pass has been defined, it will be carried out with tool radius compensation and at the indicated feedrate “H”.

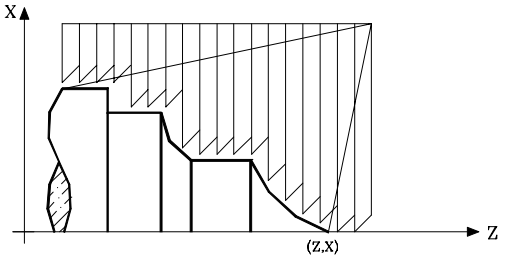
  This profile will coincide or not with the programmed profile depending on whether there are areas not accessible to the selected tool.

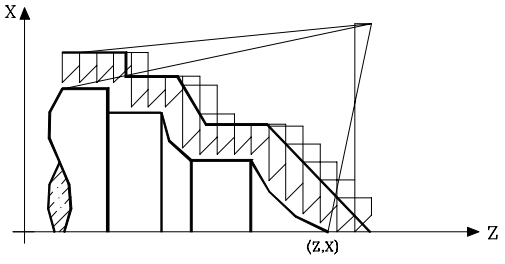


Once the finishing pass has ended, the tool will return to the point from where the cycle was called.

Optimizing the machining process

If only the desired profile is defined, the CNC assumes that the rough part is cylindrical and it machines it as shown in the drawing on the left.





When the profile of the rough part is known, it is recommended to define both profiles: the rough part and the desired final profile. The machining is faster since only the stock between both profiles is removed.

Profile programming syntax

When defining the profile, there is no need to program the initial point since it is already indicated by cycle defining parameters X and Z.

When defining 2 profiles, the final profile must be defined first and then that of the rough part.

The first and last profile defining blocks must have a label number. These labels will indicate the beginning and end of the profile geometry definition for this canned cycle.

The profile programming syntax must follow these rules:

1.- It may be programmed in absolute or incremental coordinates and it may consist of simple geometrical elements such as straight and curved lines, roundings and chamfers which will be programmed following their own syntax rules.

2.- Function G00 indicates that the profile definition is done and this block is the beginning of the definition of the rough part profile.

Program G01, G02 or G03 in the next block, because G00 is modal, thus preventing the CNC from displaying the corresponding error message.

3.- The profile description must not contain: mirror images, scaling factor changes, pattern rotation or zero offsets.

4.- It must not contain high level blocks such as jumps, calls to subroutines or parametric programming.

5.- It must not contain other canned cycles.

Only the following G functions may be programmed in the profile definition:

G00 Positioning in rapid
G01 Linear interpolation
G02 Clockwise circular interpolation
G03 Counter-clockwise circular interpolation
G06 Circle center in absolute coordinates
G08 Arc tangent to previous path
G09 Arc defined by three points
G36 Automatic radius blend
G39 Chamfer
G53 Programming with respect to machine zero
G70 Programming in inches
G71 Programming in millimeters
G90 Absolute programming
G91 Incremental programming
G93 Polar origin preset

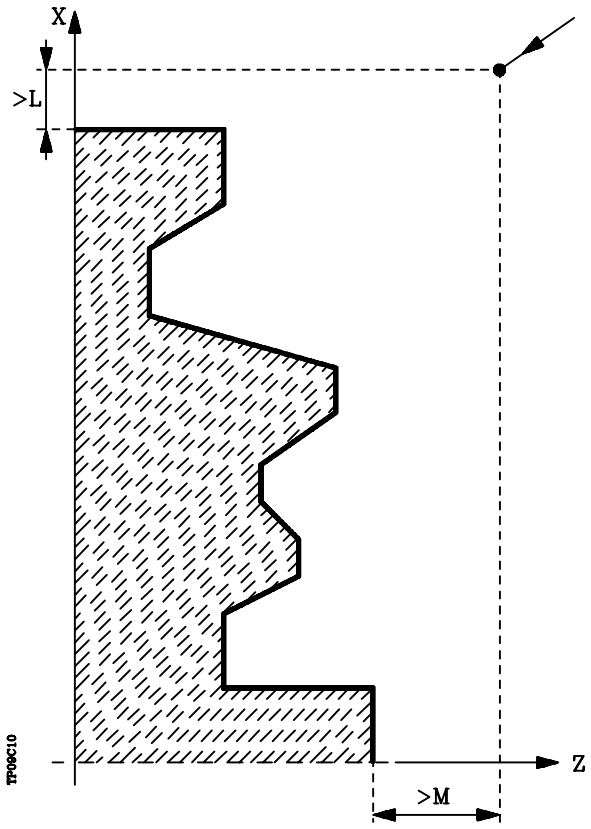
It is also possible to program the following functions although they will be ignored by the canned cycle:

G05 Round corner
G07 Square corner
G50 Controlled corner rounding
The F, S, T, D and M functions.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the canned cycle.

The canned cycle calling position must be off the part to be machined and at a distance greater than that defined as finishing stock (L, M) along the X and Z axes.



If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.

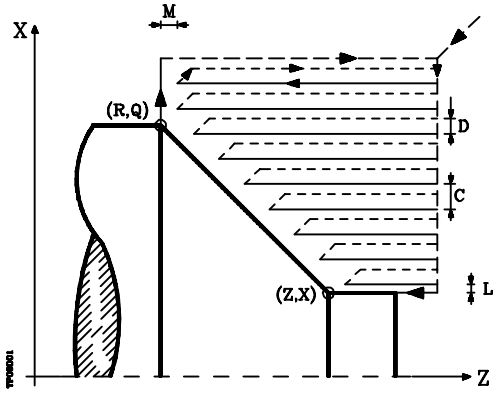
Once the canned cycle has ended, the active feedrate will be the last one programmed, i.e. the one corresponding to the roughing operation (F) or the finishing operation (H). The CNC will assume functions G00, G40 and G90.

9.4 G81. TURNING CANNED CYCLE WITH STRAIGHT SECTIONS

This cycle turns the programmed section maintaining the indicated pass between the successive turning passes.

It allows to select whether or not the canned cycle will perform a finishing pass after completing the programmed turning operation.

The basic structure of the block is: **G81 X Z Q R C D L M F H**



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

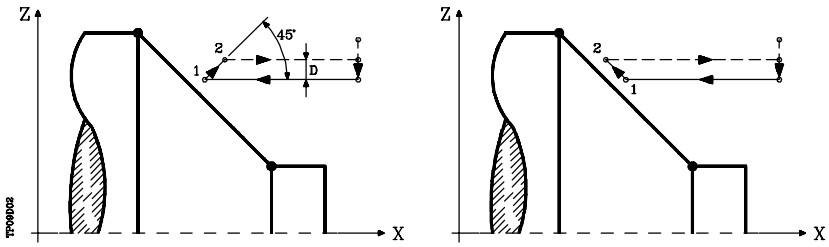
R±5.5 Defines the Z coordinate of the final point of the profile. It must be programmed in absolute values.

C5.5 Defines the turning pass. It is given by a positive radius value.

The whole turning operation is carried out with the same pass which is the same or smaller than the programmed one "C".

If programmed with a 0 value, the CNC will display the corresponding error message.

D 5.5 Defines the safety distance to withdraw the tool after each pass.



When programming "D" with a value other than "0", the cutter withdraws at a 45º angle up to the safety distance (left drawing).

When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing).

It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.

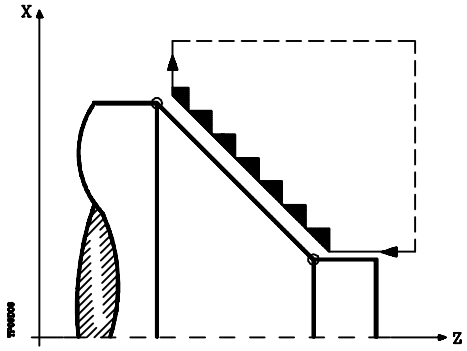
L5.5 Defines the finishing pass along X. It is given in radius.

If not programmed, a value of 0 will be assumed.

M5.5 Defines the finishing pass along Z. It is given in radius.

If not programmed, a value of 0 will be assumed.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



H5.5 Defines the feedrate for the finishing pass.

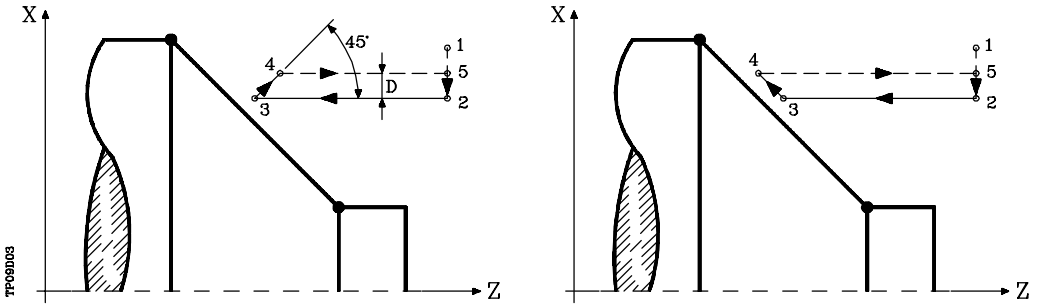
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

Basic operation:

The canned cycle will analyze the programmed profile performing, if necessary, a horizontal turning operation until the defined profile is reached.

The whole turning operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

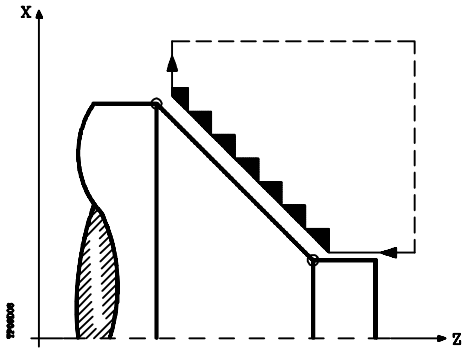
Each turning pass is carried out in the following manner:



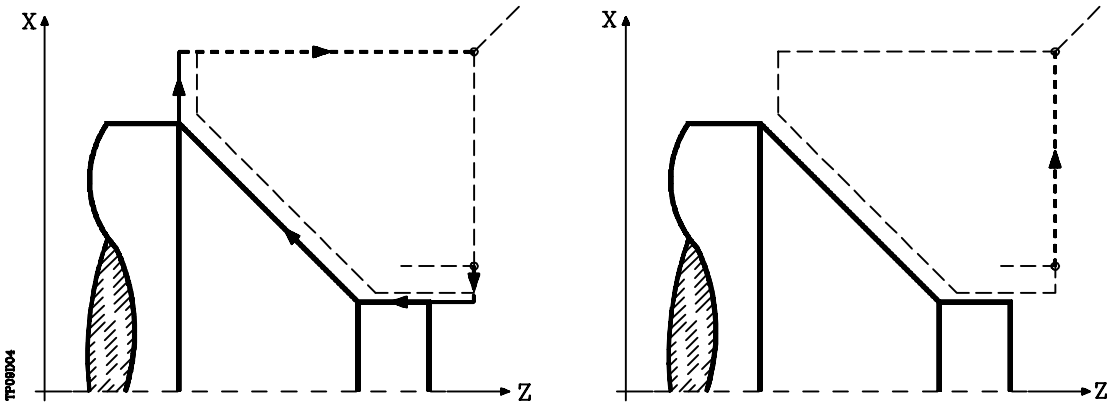
- "1-2" move in rapid (G00).
- Move "2-3" in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- Withdrawal "4-5" in rapid (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F".

This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the turning operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



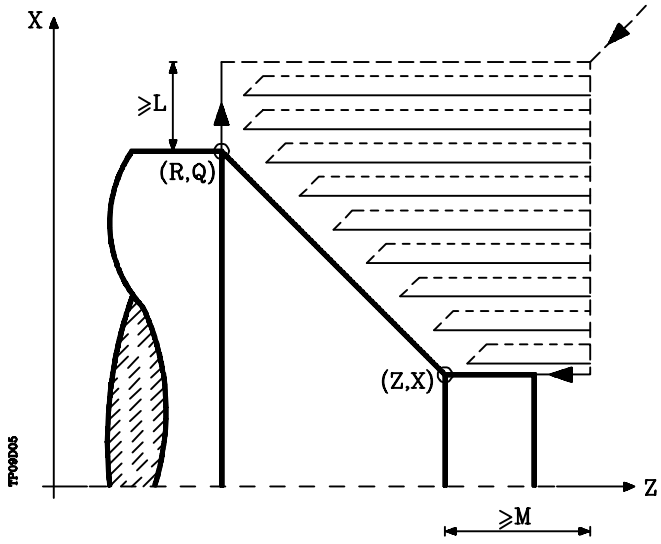
Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

The distance between the starting point and final point (R,Q) along the X axis must be equal to or greater than L.

The distance between the starting point and the initial point (Z,X) along the Z axis must be equal to or greater than M.



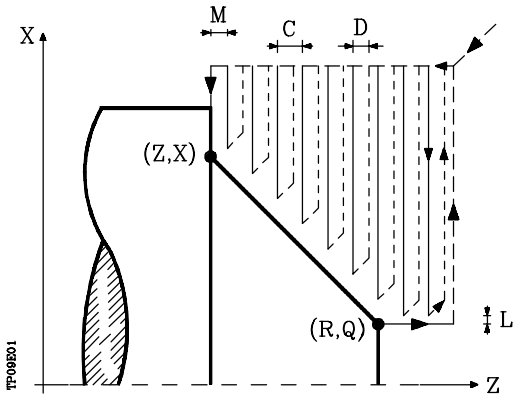
If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.

9.5 G82. FACING CANNED CYCLE WITH STRAIGHT SECTIONS

This cycle faces the programmed section maintaining the indicated pass between the successive facing passes.

It allows to select whether or not the canned cycle will perform a finishing pass after completing the programmed turning operation.

The basic structure of the block is: **G82 X Z Q R C D L M F H**



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

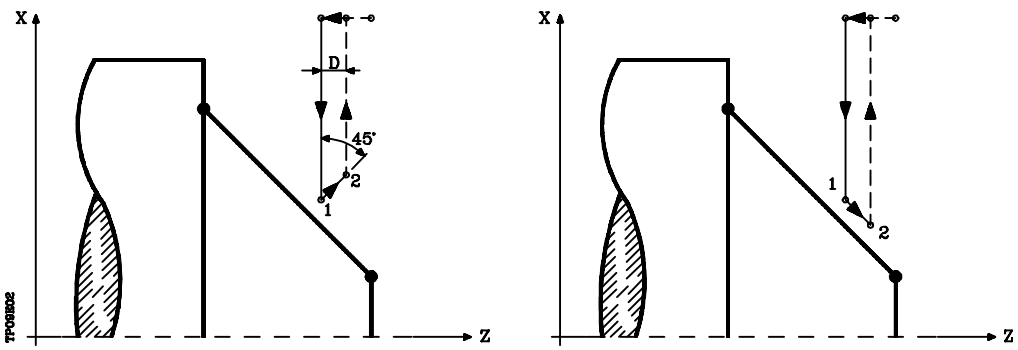
R±5.5 Defines the Z coordinate of the final point of the profile. It must be programmed in absolute values.

C5.5 Defines the facing pass.

The whole facing operation is carried out with the same pass which is the same or smaller than the one programmed "C".

If programmed as "0", the CNC will display the corresponding error.

D 5.5 Defines the safety distance to withdraw the tool after each pass.



When programming "D" with a value other than "0", the cutter withdraws at a 45º angle up to the safety distance (left drawing).

When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

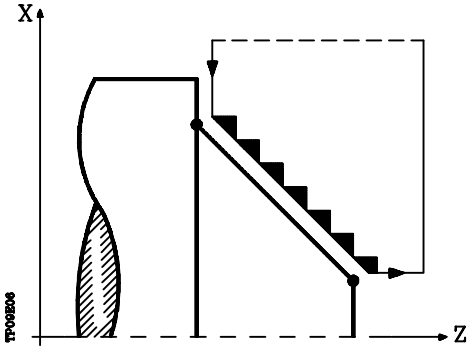
When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing).

It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.

L5.5 Defines the finishing pass along X. It is given in radius and, if not programmed, a value of 0 will be assumed.

M5.5 Defines the finishing pass along Z. It is given in radius and, if not programmed, a value of 0 will be assumed.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



H5.5 Defines the feedrate for the finishing pass.

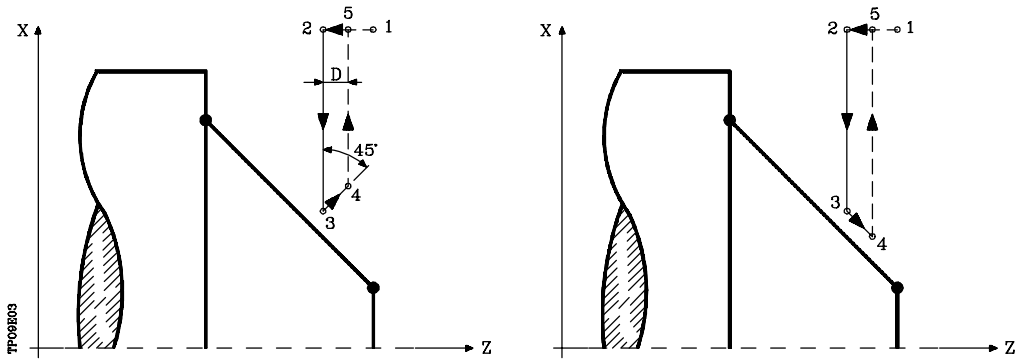
If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

Basic operation:

The canned cycle will analyze the programmed profile performing, if necessary, a facing operation until the defined profile is reached.

The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

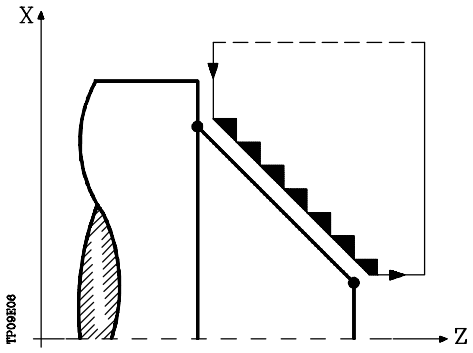
Each facing pass is carried out in the following manner:



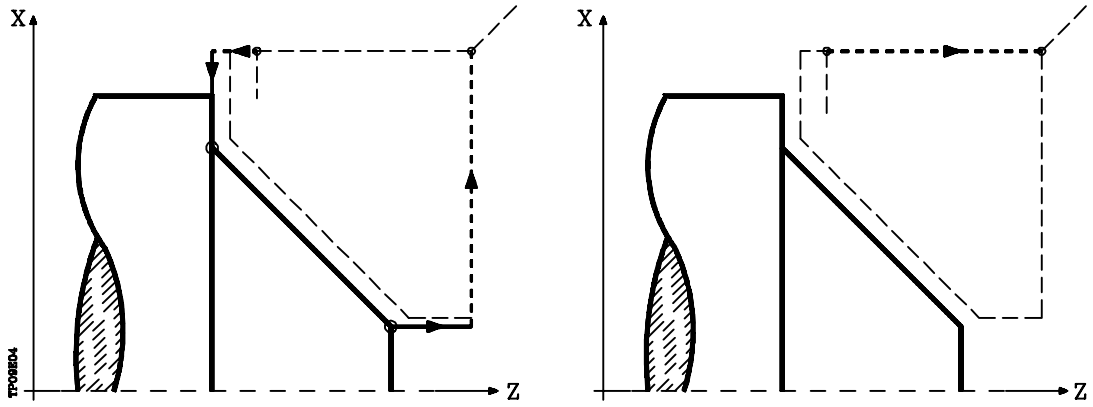
- Move “1-2” in rapid (G00).
- Move “2-3” in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- Withdrawal “4-5” in rapid (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F".

This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the facing operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



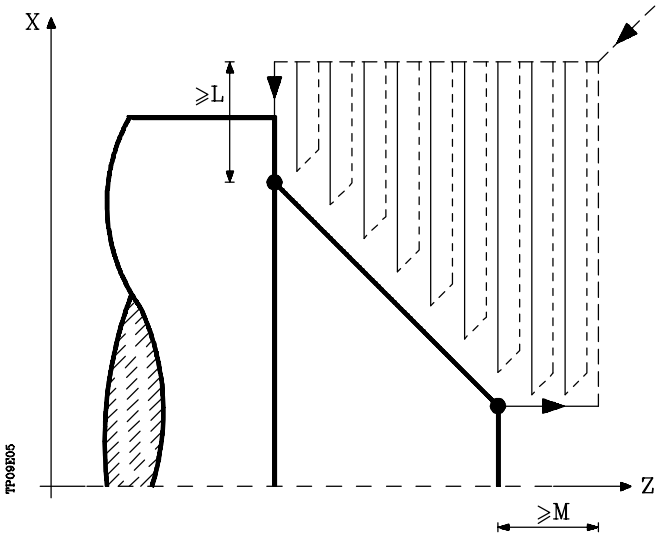
Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

The distance between the starting point and final point (Z,X) along the X axis must be equal to or greater than L.

The distance between the starting point and the initial point (R,Q) along the Z axis must be equal to or greater than M.

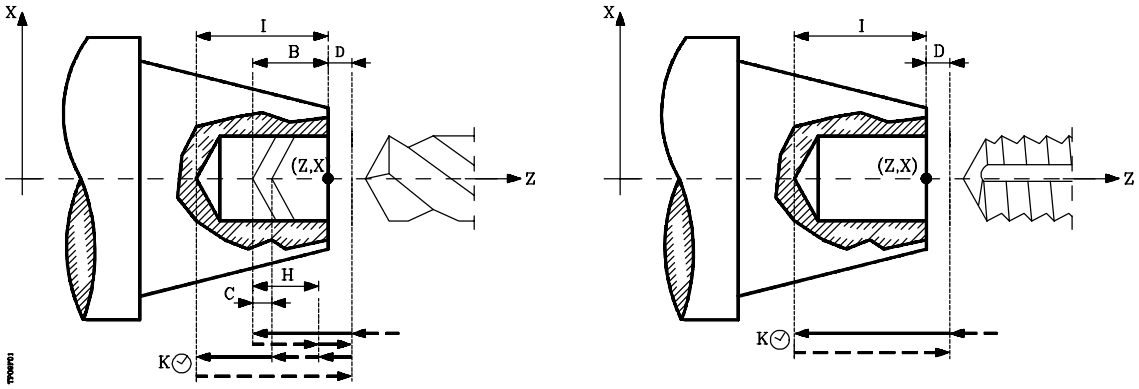


If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.

9.6 G83. AXIAL DRILLING AND TAPPING CANNED CYCLE

With this cycle, it is possible to perform an axial drilling or tapping whose selection depends on the programming format being used.

For tapping, set parameter "B=0", and for axial drilling, set "B>0".



The basic block structure for each operation is:

Axial drilling	**G83 X Z I B D K H C**
Axial tapping	**G83 X Z I B0 D K**

X±5.5 Defines the X coordinate of the drilling position. It must be programmed in absolute and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the drilling position. It must be programmed in absolute.

I±5.5 Defines the total drilling depth. It is referred to the drilling point. Thus, it will have a positive value when drilling in the negative Z direction and vice versa.

If programmed with a value of 0, the CNC will display the corresponding error message.

B5.5 Defines the type of operation to be performed
* If B=0, the CNC will carry out an axial tapping.
* If B>0 it will carry out an axial drilling where B indicates the drilling step.

D5.5 Defines the safety distance and it indicates at what distance the tool is positioned in the approach move. If not programmed, a value of 0 is assumed.

K5 Defines the dwell, in hundredths of a second, at the bottom of the hole before starting the withdrawal. If not programmed, a value of 0 is assumed.

H5.5 Defines the withdrawal distance after each drilling step (in G00). If not programmed or programmed a value of 0, it will withdraw to the approach point.

C5.5 Defines the approach distance along Z from the previous drilling step to start the next drilling step (in G00).

If not programmed, a value of 1 millimeter.

Drilling. Basic operation:

1.- Rapid move (in G00) to the approach point located at a distance “D” from the the drilling point.

2.- First drilling operation. The longitudinal axis moves in G1 at programmed feedrate to the programmed incremental depth “D+B”.

3.- Drilling loop. The following steps will be repeated until the total programmed depth “I” is reached.

3.1.- Withdrawal in rapid (G00) a distance “H” or to the approach point.

3.2.- Rapid approach (G00) up to a distance “C” from the previous drilling step.

3.3.- New drilling step. G1 move to the next incremental depth “B”.

4.- Dwell “K” in hundredths of a second at the bottom of the hole, if programmed.

5.- Withdrawal in rapid (G00) to the approach point.

Tapping. Basic operation:

1.- Rapid move (in G00) to the approach point located at a distance “D” from the the tapping point.

2.- Tapping, the longitudinal axis moves in G01 to the programmed depth “D+I”.

3.- The spindle reverses turning direction.

If "K" has been programmed, the spindle stops and after the programmed dwell has elapsed, the spindle starts turning in the other way.

4.- Withdrawl in G01 to the approach point.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

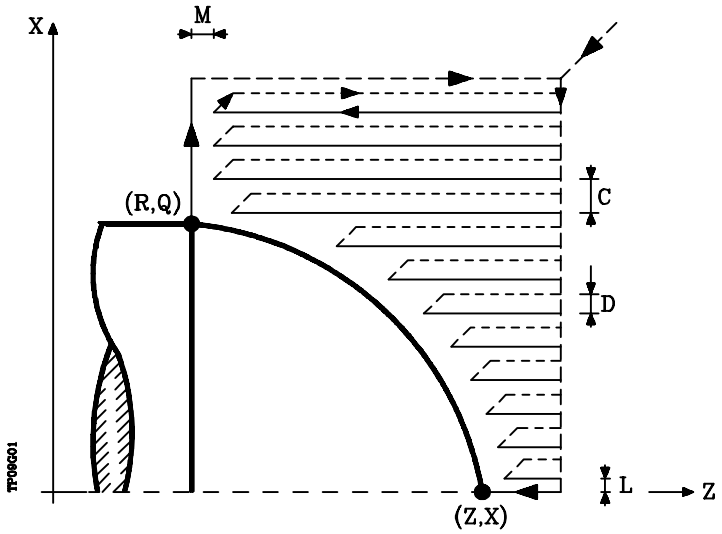
Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

9.7 G84. TURNING CANNED CYCLE WITH ARCS

This cycle turns the programmed section maintaining the indicated pass between the successive turning passes.

It allows to select whether or not the canned cycle will perform a finishing pass after completing the programmed turning operation.

The basic structure of the block is: G84 X Z Q R C D L M F H I K



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter)

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

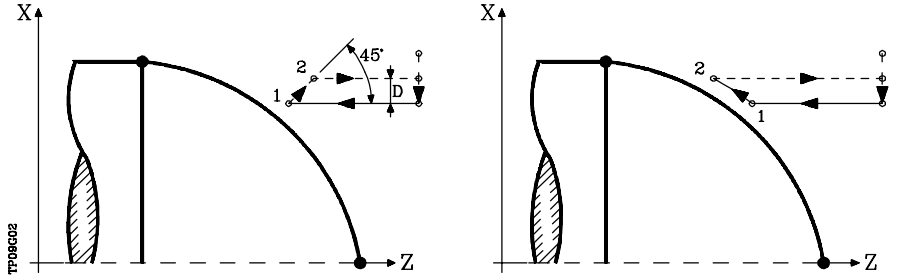
R±5.5 Defines the Z coordinate of the final point of the profile. It must be programmed in absolute values.

C5.5 Defines the turning pass. It is given by a positive radius value.

The whole turning operation is carried out with the same pass value which will be equal to or smaller than the one programmed "C".

If programmed with a 0 value, the CNC will display the corresponding error message.

D 5.5 Defines the safety distance to withdraw the tool after each pass.



When programming "D" with a value other than "0", the cutter withdraws at a 45º angle up to the safety distance (left drawing).

When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

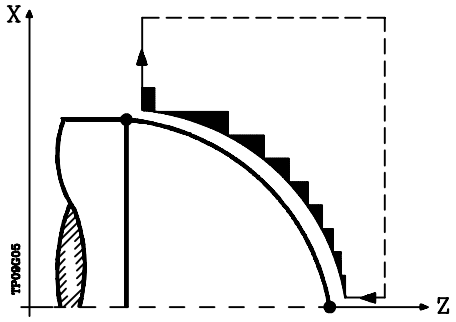
When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing).

It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.

L5.5 Defines the finishing pass along X. It is given in radius and, if not programmed, a value of 0 will be assumed.

M5.5 Defines the finishing pass along Z. It is given in radius and, if not programmed, a value of 0 will be assumed.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



H5.5 Defines the feedrate for the finishing pass.

If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

I±5.5 Defines the distance, in radius, from the initial point (X, Z) to the arc's center along the X axis. It is programmed in incremental values with respect to the initial point like the "I" for circular interpolations (G02, G03).

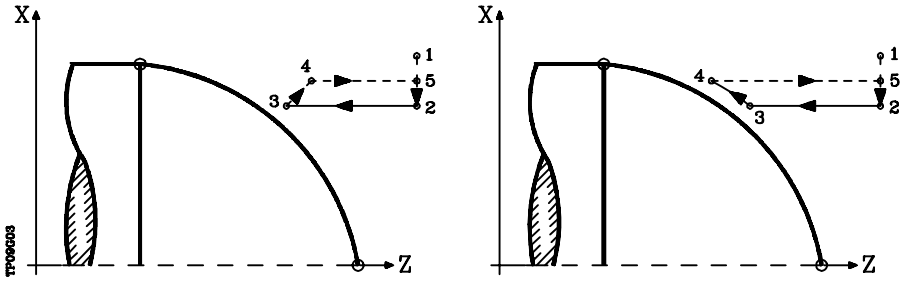
K±5.5 Defines the distance from the initial point (X, Z) to the arc's center along the Z axis. It is programmed in incremental values with respect to the initial point like the "K" for circular interpolations (G02, G03).

Basic operation:

The canned cycle will analyze the programmed profile performing, if necessary, a horizontal turning operation until the defined profile is reached.

The whole turning operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

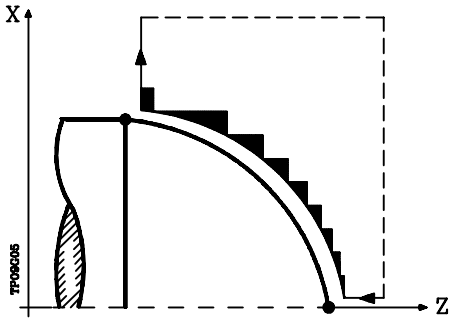
Each turning pass is carried out in the following manner:



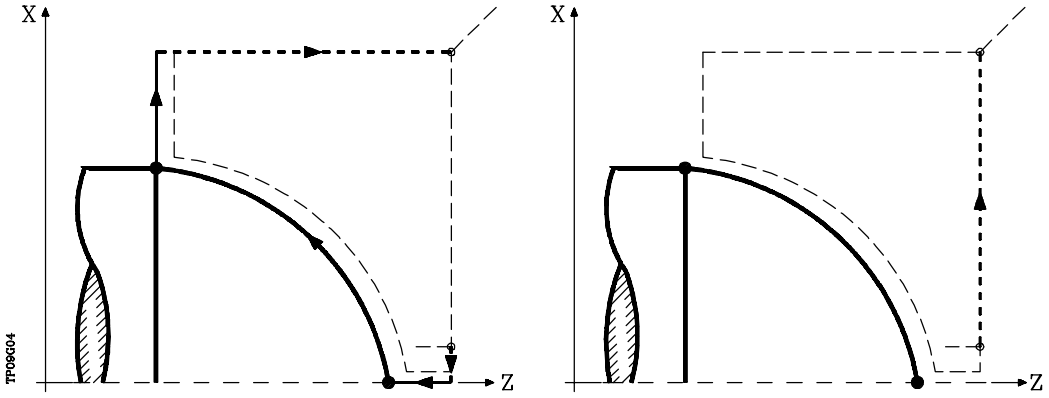
- Move “1-2” in rapid (G00).
- Move “2-3” in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- Withdrawal “4-5” in rapid (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F".

This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the turning operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



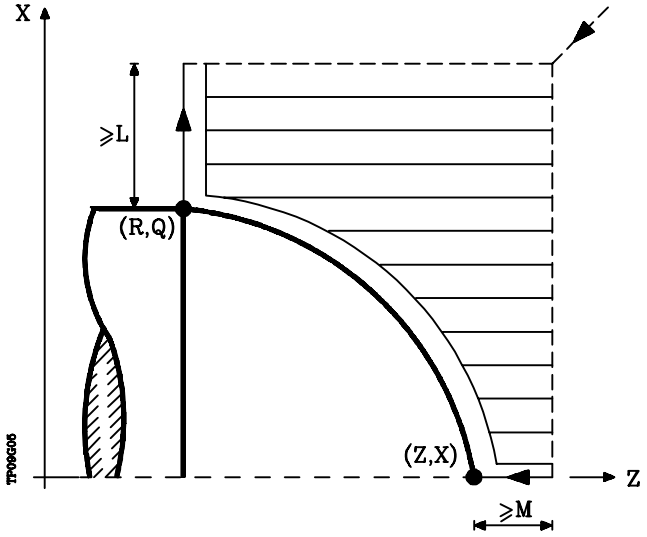
Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

The distance between the starting point and final point (R,Q) along the X axis must be equal to or greater than L.

The distance between the starting point and the initial point (Z,X) along the Z axis must be equal to or greater than M.



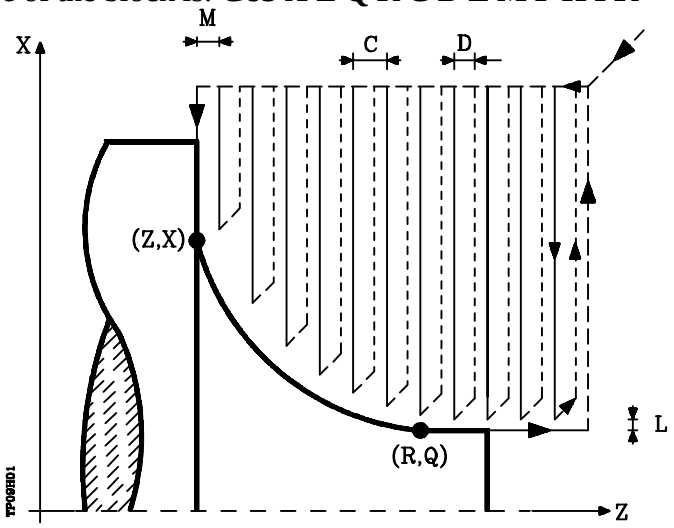
If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.

9.8 G85. FACING CANNED CYCLE WITH ARCS

This cycle faces the programmed section maintaining the indicated pass between the successive facing passes.

It allows to select whether or not the canned cycle will perform a finishing pass after completing the programmed facing operation.

The basic structure of the block is: **G85 X Z Q R C D L M F H I K**



X±5.5 Defines the X coordinate of the initial point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the profile. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the profile. It must be programmed in absolute values and in the active units (radius or diameter).

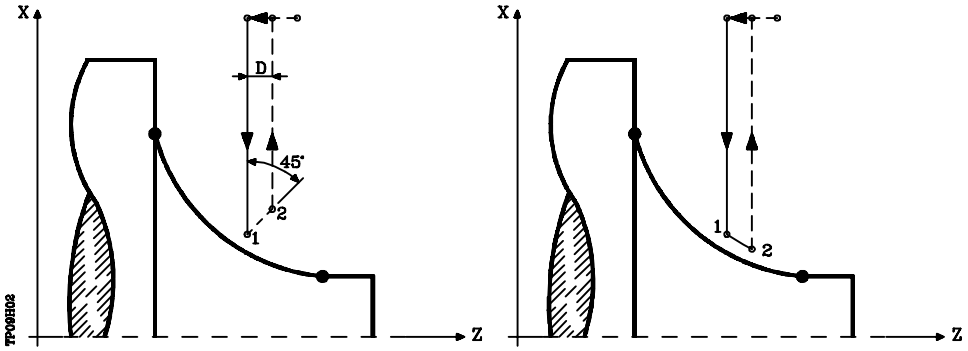
R±5.5 Defines the Z coordinate of the final point of the profile. It must be programmed in absolute values.

C5.5 Defines the facing pass.

The whole facing operation is carried out with the same pass value which will be equal to or smaller than the one programmed "C".

If programmed with a 0 value, the CNC will display the corresponding error message.

D 5.5 Defines the safety distance to withdraw the tool after each pass.



When programming "D" with a value other than "0", the cutter withdraws at a 45º angle up to the safety distance (left drawing).

When programming "D" with a "0" value, the exit path is the same as the entry path. This may be interesting for grooving complex profiles, to use these cycles on cylindrical grinders, etc.

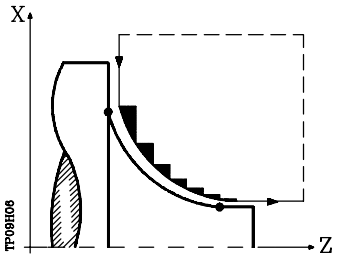
When not programming "D", the tool withdrawal is carried out following the profile up to the last pass, "C" distance (right drawing).

It must be borne in mind that when "D" is not programmed, the cycle execution time is greater; but the amount of stock to be removed on the finishing pass is less.

L5.5 Defines the finishing pass along X. It is given in radius and, if not programmed, a value of 0 will be assumed.

M5.5 Defines the finishing pass along Z. It is given in radius and, if not programmed, a value of 0 will be assumed.

F5.5 Defines the feedrate for the final roughing pass. If not programmed or programmed with a value of "0", it means that there is no final roughing pass.



H5.5 Defines the feedrate for the finishing pass.

If not programmed or programmed with a value of 0, it is considered that no finishing pass is desired.

I±5.5 Defines the distance, in radius, from the initial point (X, Z) to the arc's center along the X axis. It is programmed in incremental values with respect to the initial point like the "I" for circular interpolations (G02, G03).

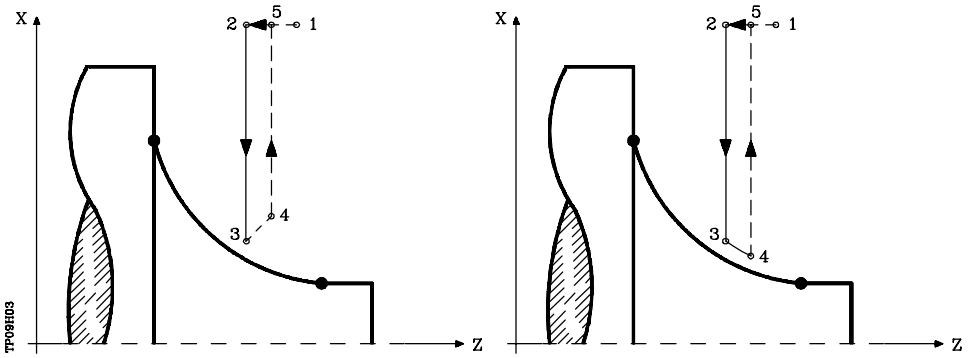
K±5.5 Defines the distance from the initial point (X, Z) to the arc's center along the Z axis. It is programmed in incremental values with respect to the initial point like the "K" for circular interpolations (G02, G03).

Basic operation:

The canned cycle will analyze the programmed profile performing, if necessary, a facing operation until the defined profile is reached.

The whole facing operation is carried out with the same pass, this being equal to or smaller than the programmed one (C).

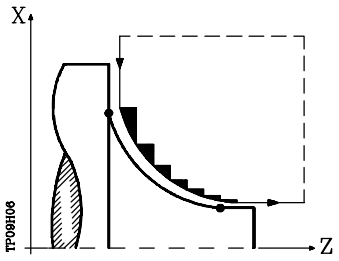
Each facing pass is carried out in the following manner:



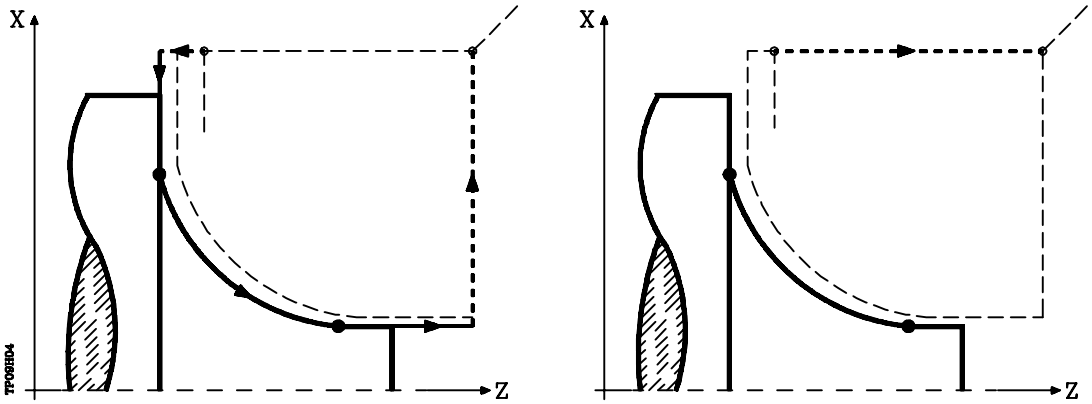
- Move “1-2” in rapid (G00).
- Move “2-3” in G01 at programmed feedrate (F).
- If "D" has been programmed, the 3-4 move is done in rapid (G00); but if "D" has not been programmed, the 3-4 move is done following the profile in G01 at the programmed feedrate (F).
- Withdrawal “4-5” in rapid (G00).

If a final roughing pass has been programmed, the CNC will run a pass parallel to the profile maintaining an "L" and "M" stock at the set feedrate "F".

This final roughing pass will eliminate the ridges that were left in the roughing stage.



After the facing operation (with or without finishing pass) the canned cycle will always end at the cycle calling point.



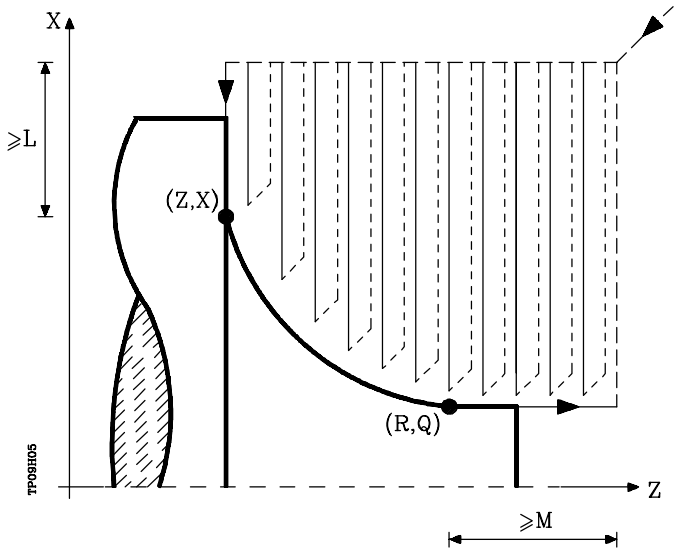
Considerations

The machining conditions (feedrate, spindle speed, etc.) as well as the tool radius compensation (G41, G42) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

The distance between the starting point and final point (Z, X) along the X axis must be equal to or greater than L.

The distance between the starting point and the initial point (R,Q) along the Z axis must be equal to or greater than M.

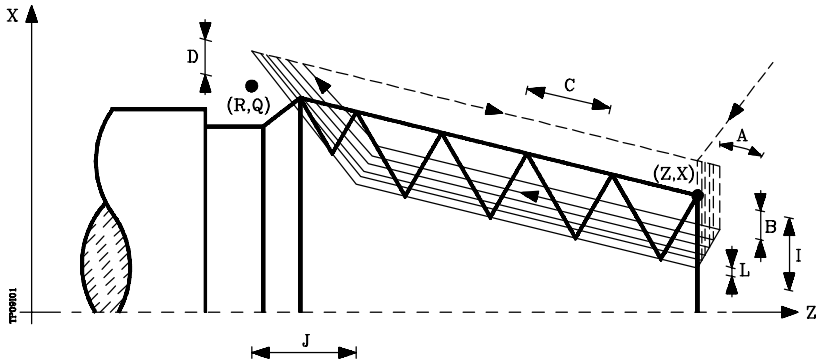


If the tool position is not correct to execute the cycle, the CNC will display the corresponding error message.

9.9 G86. LONGITUDINAL THREADCUTTING CANNED CYCLE

With this cycle it is possible to cut outside, inside and tapered threads of constant pitch.

The basic block structure is: **G86 X Z Q R K I B E D L C J A W**



X±5.5 Defines the X coordinate of the initial point of the thread. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the thread. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the thread. It must be programmed in absolute and in radius or diameter depending on the active units.

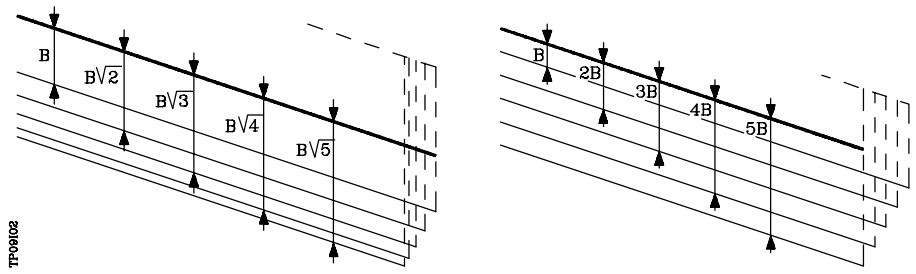
R±5.5 Defines the Z coordinate of the final point of the thread.

K±5.5 Optional. It is used with parameter "W" for thread repair. Defines the Z coordinate of the point where the thread is to be measured. Usually it is an intermediate point of the thread.

I±5.5 Defines the depth of the thread in radius. Its value must be positive for outside threads and negative for inside ones.

If a value of 0 is programmed, the CNC will display the corresponding error message.

B±5.5 Defines the depth of the threading passes and it is given in radius.



* If a positive value is programmed, the depth of each pass will depend on the number of the corresponding pass.

  This way, the penetrations along the X axis are:

  $$B,\ B\sqrt{2},\ B\sqrt{3},\ B\sqrt{4},\ \ldots..\ B\sqrt{n}$$

* If a negative value is programmed, the passes will have the same depth with a value equal to the one programmed (B).

  This way the penetrations along X axis are:

  B,2B, 3B, 4B, ... nB

* If a value of 0 is programmed, the CNC will display the corresponding error message.

E±5.5 It is related to parameter "B".
It indicates the minimum value to be reached the penetration pitch when parameter "B" has been programmed with a positive value.

If not programmed, a value of "0" will be assumed.

D±5.5 Defines the safety distance, along X, and it indicates at what distance from the initial point must the tool be positioned in the approach move. It is programmed in radius.

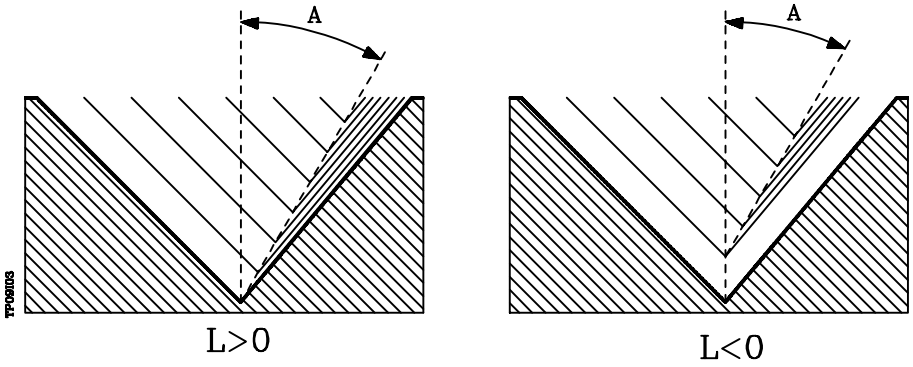
The return to the initial point after each pass will be carried out maintaining this distance (D) from the programmed section.

If the programmed value is positive, this withdrawal will be performed in round corner (G05) and if negative, in square corner (G07).

If not programmed, a value of 0 will be assumed.

L±5.5 Defines the finishing pass and it is programmed in radius.

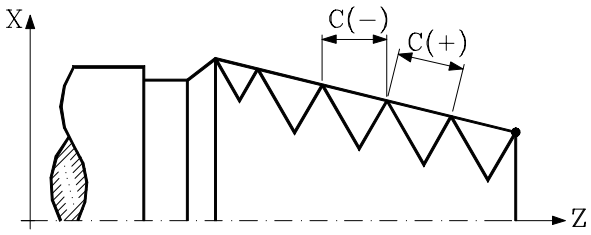
* If a positive value is programmed, the finishing pass will be performed maintaining the same penetration angle “A” as for the other passes.



* If a negative value is programmed, the finishing pass will be carried out with radial penetration.

* If a value of 0 is programmed, the previous pass is repeated.

C5.5 Defines the thread's pitch.



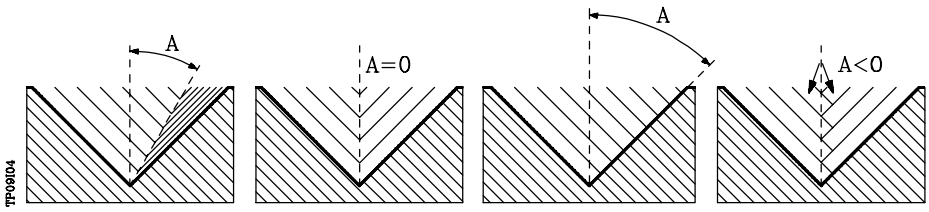
With a positive sign if the pitch is programmed along the taper
With a negative sign if the pitch is programmed along the associated axis.
If a value of 0 is programmed, the CNC will display the corresponding error message.

The right-hand and left-hand threads will be programmed by indicating the direction of the spindle rotation M03 or M04.

J5.5 Exit from the thread. It defines at what distance from the final point (R,Q), along the Z axis, starts the withdrawal from the thread.

If not programmed, a value of 0 is assumed.

A±5.5 Defines the penetration angle of the tool. It is referred to X axis and if not programmed, a value of 30° will be taken.



If A=0 is programmed, the thread will be made with radial penetration.

If the value assigned to parameter "A" is half the tool angle, the penetration will be done rubbing the thread's flank.

If A is programmed with a negative value, the penetration will be done alternating the flanks of the thread.

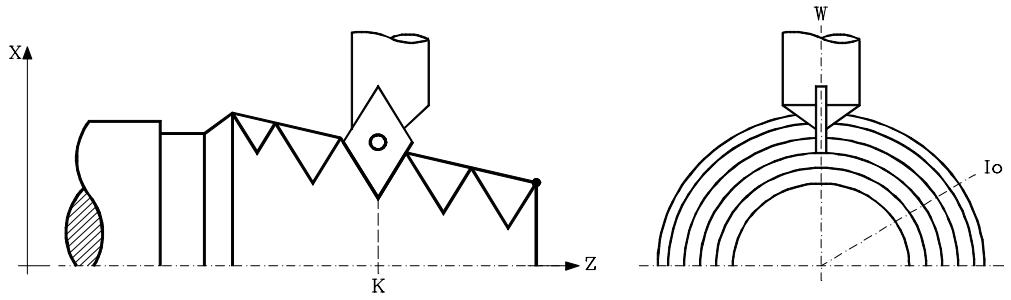
W±5.5 Optional. Its meaning depends on parameter "K".

If parameter "K" has not been defined, it indicates the angular position of the spindle corresponding to the thread's starting point. With this, it is possible to make multiple entry threads.

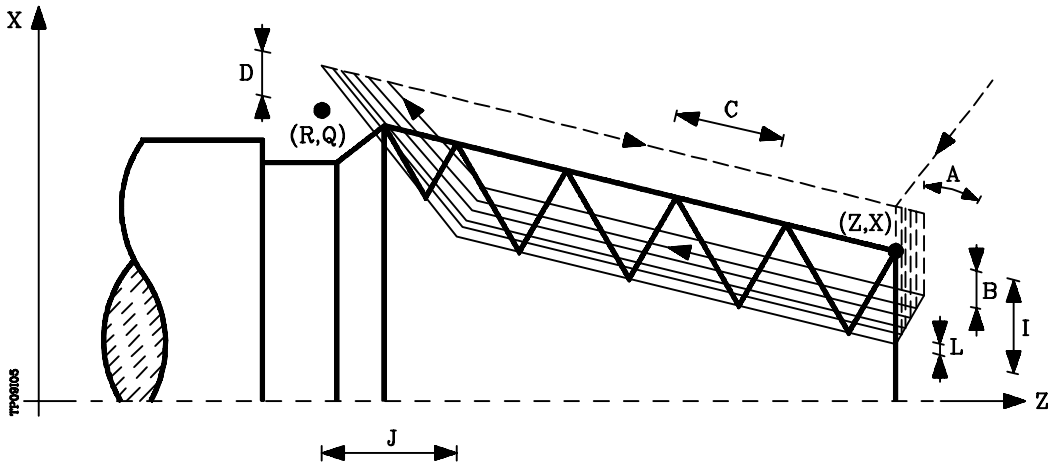
The following example shows how to make a 3-entry-point thread. Ti do that, 3 threading canned cycles will be programmed with the same values except for "W".

```
G86 X Z Q R K I B E D L C J A W0
G86 X Z Q R K I B E D L C J A W120
G86 X Z Q R K I B E D L C J A W240
```

If "K" has been defined, it is a thread repair cycle. It indicates the angular spindle position corresponding to the thread measuring point.



Basic operation:



1.- Rapid move (G00) to the approach point located at a safety distance "D" from the initial point (Z,X).

2.- Threading loop. The following steps will be repeated until the finishing coordinate is reached; programmed depth "I" minus the finishing stock "L".

2.1.- Rapid move (G00) to the programmed depth indicated by "B".

This move will be made according to the penetration angle of the selected tool (A).

2.2.- Performs the threading of the programmed section with the selected exit "J". Neither can the feedrate F be altered with the Feedrate Override Switch nor the spindle speed S by means of the spindle speed override keys while threading. They will be kept at 100%.

2.3.- Withdrawal in rapid (G00) to the approach point.

3.- Thread finishing. Rapid move (G00) to the programmed depth coordinate "I".

This move will be radial or according to the tool penetration angle (A) which depends on the sign assigned to parameter "L".

4.- It threads the programmed section with the selected exit (J).

Neither can the feedrate F be altered with the Feedrate Override Switch nor the spindle speed S by means of the spindle speed override keys while threading. They will be kept at 100%.

5.- Withdrawal in rapid (G00) to the approach point.

Considerations

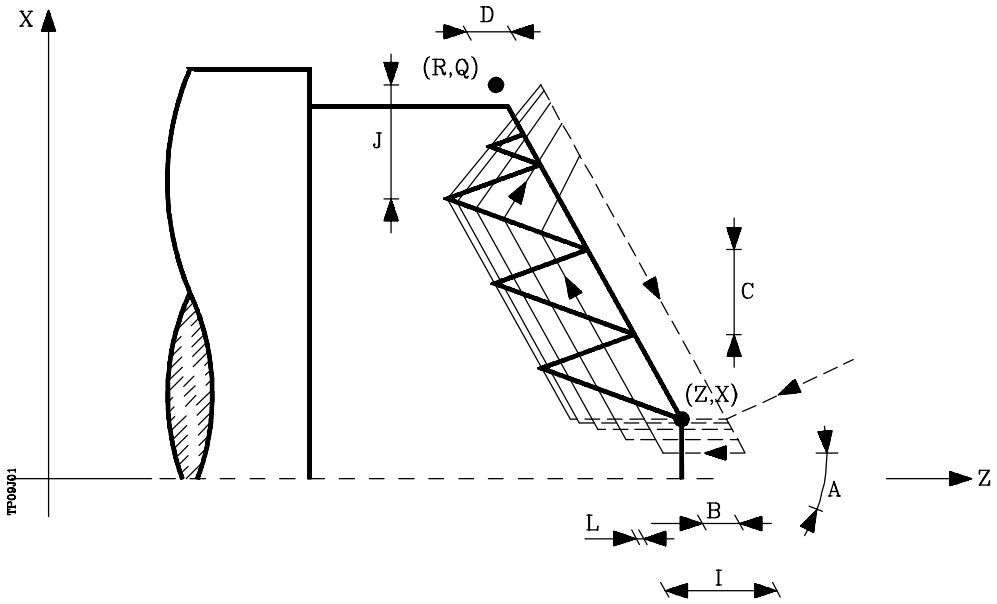
The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

9.10 G87. FACE THREADCUTTING CANNED CYCLE

With this cycle it is possible to cut outside and inside threads of constant face pitch.

The basic block structure is: **G87 X Z Q R K I B E D L C J A W**



X±5.5 Defines the X coordinate of the initial point of the thread. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the thread. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the thread. It must be programmed in absolute and in radius or diameter depending on the active units.

R±5.5 Defines the Z coordinate of the final point of the thread.

K±5.5 Optional. It is used with parameter "W" for thread repair. Defines the Z coordinate of the point where the thread is to be measured. Usually it is an intermediate point of the thread.

I±5.5 Defines the depth of the thread. Its value will be positive if threading in the negative Z direction and vice versa.

If a value of 0 is programmed, the CNC will display the corresponding error message.

B±5.5 Defines the depth of the threading passes and it is given in radius.

* If a positive value is programmed, the depth of each pass will depend on the number of the corresponding pass.

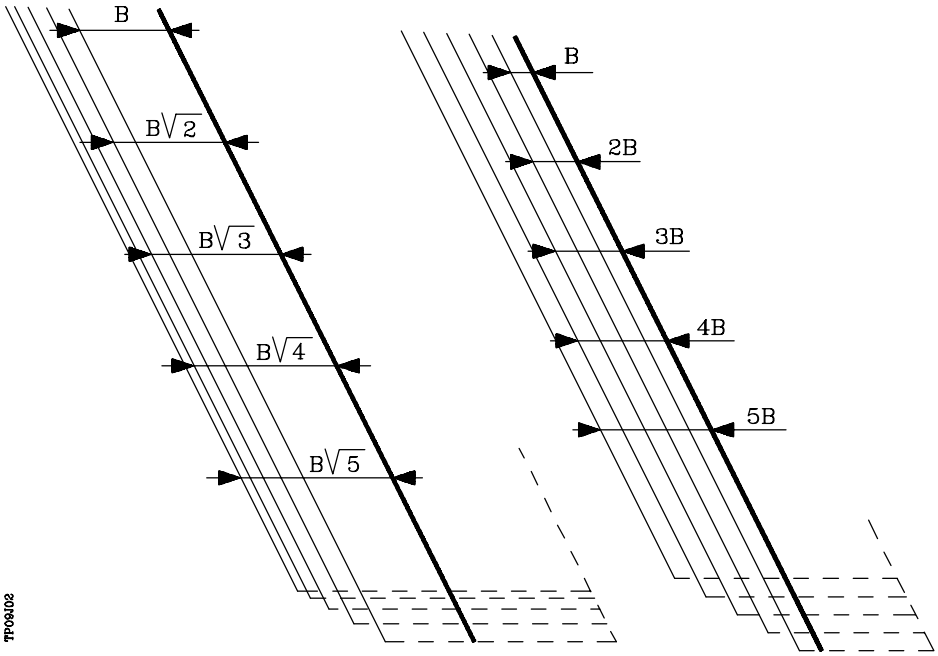
This way, the penetrations along the Z axis are:

$$B,\ B\sqrt{2},\ B\sqrt{3},\ B\sqrt{4},\ \ldots..\ B\sqrt{n}$$

* If a negative value is programmed, the passes will have the same depth with a value equal to the one programmed (B).

  This way, the penetrations along the Z axis are:

  B,2B, 3B, 4B, ... nB



* If a value of 0 is programmed, the CNC will display the corresponding error message.

E±5.5 It is related to parameter "B".
It indicates the minimum value to be reached the penetration pitch when parameter "B" has been programmed with a positive value.

If not programmed, a value of "0" will be assumed.

D±5.5 Defines the safety distance and it indicates at what distance, along Z, from the initial point must the tool be positioned in the approach move. It is programmed in radius.

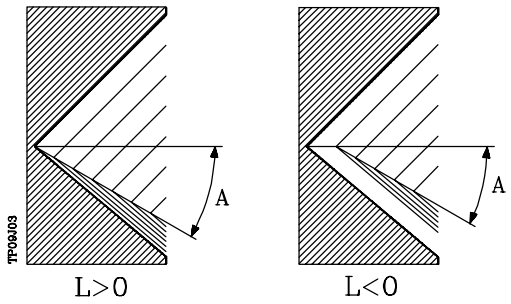
The return to the initial point after each pass will be carried out maintaining this distance (D) from the programmed section.

If the programmed value is positive, this withdrawal will be performed in round corner (G05) and if negative, in square corner (G07).

If not programmed, a value of 0 will be assumed.

L±5.5 Defines the finishing pass

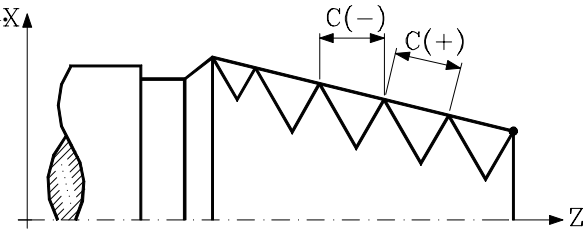
* If a positive value is programmed, the finishing pass will be performed maintaining the same penetration angle "A" as for the other passes.



* If a negative value is programmed, the finishing pass will be carried out with radial penetration.

* If a value of 0 is programmed, the previous pass is repeated.

C5.5 Defines the thread's pitch



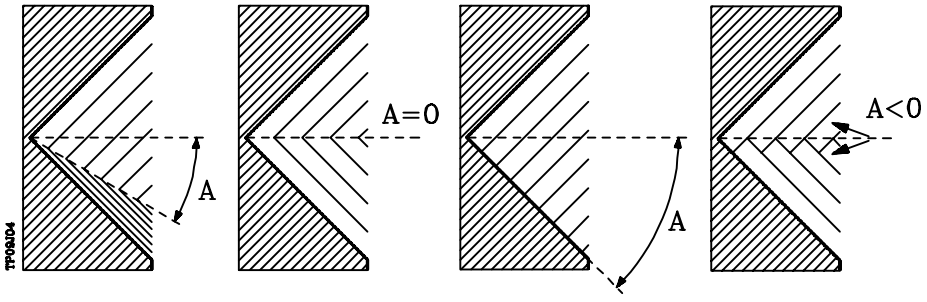
With a positive sign if the pitch is programmed along the taper
With a negative sign if the pitch is programmed along the associated axis.
If a value of 0 is programmed, the CNC will display the corresponding error message.

The right-hand and left-hand threads will be programmed by indicating the direction of the spindle rotation M03 or M04.

J5.5 Exit from the thread. It defines at what distance from the final point (R,Q), along the X axis, starts the withdrawal from the thread.

If not programmed, a value of 0 is assumed.

A±5.5 Defines the penetration angle of the tool. It is referred to the X axis and if not programmed, a value of 30o will be taken.



If A=0 is programmed, the thread will be made with radial penetration.

If the value assigned to parameter "A" is half the tool angle, the penetration will be done rubbing the thread's flank.

If A is programmed with a negative value, the penetration will be done alternating the flanks of the thread.

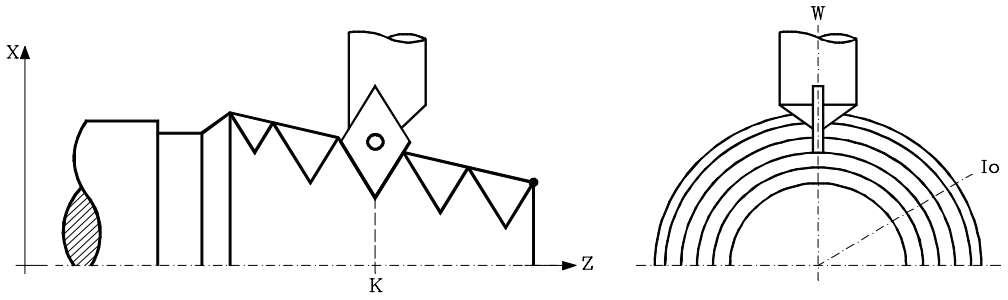
W±5.5 Optional. Its meaning depends on parameter "K".

If parameter "K" has not been defined, it indicates the angular position of the spindle corresponding to the thread's starting point. With this, it is possible to make multiple entry threads.

The following example shows how to make a 3-entry-point thread. To do that, 3 threading canned cycles will be programmed with the same values except for "W".

```
G86 X Z Q R K I B E D L C J A W0
G86 X Z Q R K I B E D L C J A W120
G86 X Z Q R K I B E D L C J A W240
```

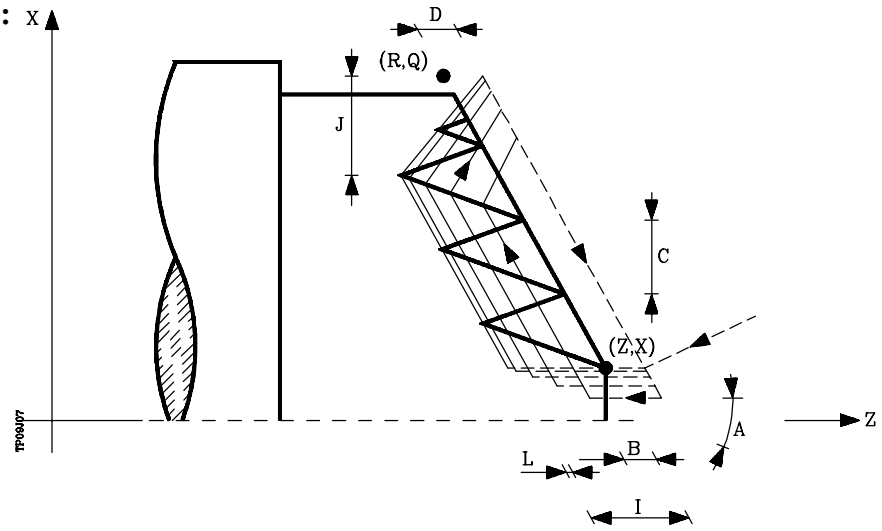
If "K" has been defined, it is a thread repair cycle. It indicates the angular spindle position corresponding to the thread measuring point.



Note: To do a thread repair, proceed as follows:

1- Home (reference) the spindle.
2- Measure the angle of the root of the thread, K W parameters.
3- Define the G87 cycle for thread repair.
4- Execute the canned cycle.

Basic operation:



1.- Rapid move (G00) to the approach point located at a safety distance "D" from the initial point (Z,X).

2.- Threading loop. The following steps will be repeated until the finishing coordinate is reached; programmed depth "I" minus the finishing stock "L".

2.1.- Rapid move (G00) to the programmed depth indicated by "B".

This move will be made according to the penetration angle of the selected tool (A).

2.2.- Performs the threading of the programmed section with the selected exit "J". Neither can the feedrate F be altered with the Feedrate Override Switch nor the spindle speed S by means of the spindle speed override keys while threading. They will be kept at 100%.

2.3.- Withdrawal in rapid (G00) to the approach point.

3.- Thread finishing. Rapid move (G00) to the programmed depth coordinate "I".

This move will be radial or according to the tool penetration angle (A) which depends on the sign assigned to parameter "L".

4.- It threads the programmed section with the selected exit (J).

Neither can the feedrate F be altered with the Feedrate Override Switch nor the spindle speed S by means of the spindle speed override keys while threading. They will be kept at 100%.

5.- Withdrawal in rapid (G00) to the approach point.

Considerations

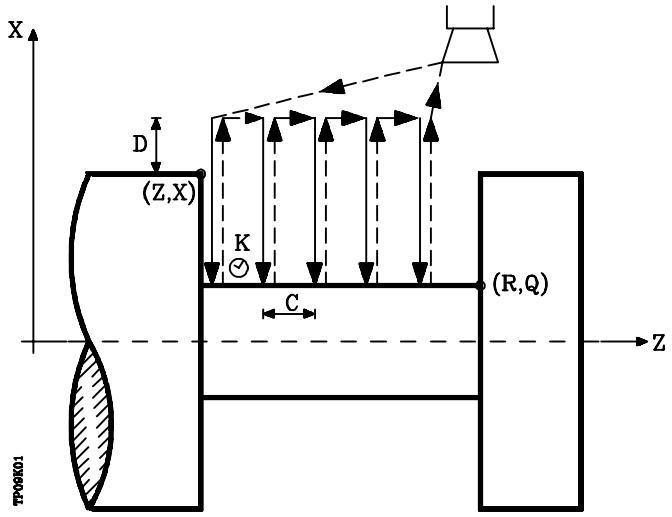
The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

9.11 G88. GROOVING CANNED CYCLE ALONG THE X AXIS

This cycle performs a grooving operation along the X axis maintaining the same pass between the successive passes being this equal to or smaller than the one programmed.

The basic structure of the block is: **G88 X Z Q R C D K**



X±5.5 Defines the X coordinate of the initial point of the groove. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the groove. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the groove. It must be programmed in absolute and in radius or diameter depending on the active units.

R±5.5 Defines the Z coordinate of the final point of the groove.

C5.5 Defines the grooving pass.

If not programmed, the CNC will assume the width of the cuttter (NOSEW) of the active tool and if programmed with a zero value, the CNC will issue the corresponding error message.

D5.5 Defines the safety distance and it must have a positive value in radius.

If not programmed, a value of 0 will be assumed.

K5 Defines the dwell, in hundredths of a second, after each penetration until the withdrawal begins.

If not programmed, a value of 0 is assumed.

Basic operation:

The whole grooving operation is done with the same pass being this equal to or smaller than "C".

Each grooving pass is carried out in the following manner:

* The penetration is done at programmed feedrate (F).

* The withdrawal and the move to the next penetration point are done in rapid (G00).

After completing the grooving operation, the canned cycle will always end at the cycle calling point.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

The tool must be located at a distance, along X, equal to or greater than the one indicated by canned cycle parameter “D” (safety distance) with respect to the part.

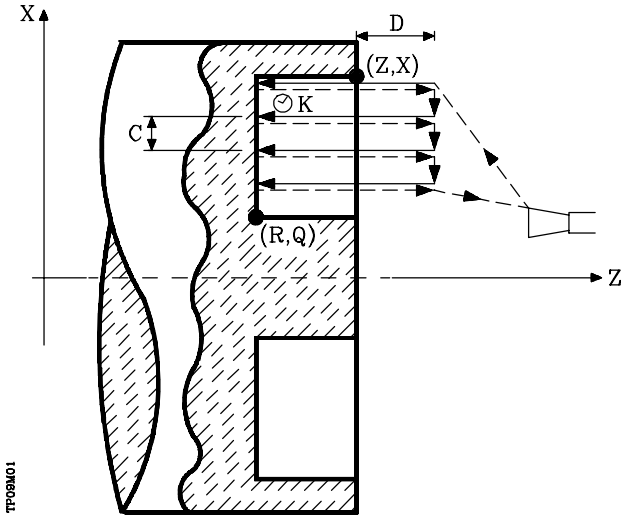
If the depth of the groove is 0, the CNC will display the corresponding error message.

If the width of the groove is smaller than the width of the cutter (NOSEW), the CNC will display the corresponding error message.

9.12 G89. GROOVING CANNED CYCLE ALONG THE Z AXIS

This cycle performs a grooving operation along the Z axis maintaining the same pass between the successive passes being this equal to or smaller than the one programamed.

The basic structure of the block is: **G89 X Z Q R C D K**



X±5.5 Defines the X coordinate of the initial point of the groove. It must be programmed in absolute values and in the active units (radius or diameter).

Z±5.5 Defines the Z coordinate of the initial point of the groove. It must be programmed in absolute values.

Q±5.5 Defines the X coordinate of the final point of the groove. It must be programmed in absolute and in radius or diameter depending on the active units.

R±5.5 Defines the Z coordinate of the final point of the groove.

C5.5 Defines the grooving pass in radius.

If not programmed, the CNC will assume the width of the cuttter (NOSEW) of the active tool and if programmed with a zero value, the CNC will issue the corresponding error message.

D5.5 Defines the safety distance and it must have a positive value in radius.

If not programmed, a value of 0 will be assumed.

K5 Defines the dwell, in hundredths of a second, after each penetration until the withdrawal begins.

If not programmed, a value of 0 is assumed.

Basic operation:

The whole grooving operation is done with the same pass being this equal to or smaller than "C".

Each grooving pass is carried out in the following manner:

* The penetration is done at programmed feedrate (F).

* The withdrawal and the move to the next penetration point are done in rapid (G00).

After completing the grooving operation, the canned cycle will always end at the cycle calling point.

Considerations

The machining conditions (feedrate, spindle speed, etc.) must be programmed before calling the cycle.

Once the canned cycle has ended, the program will continue with the same feedrate F and G functions active previous to calling the cycle. Only the tool radius compensation will be cancelled (G40) if it was active.

The tool must be located at a distance, along Z, equal to or greater than the one indicated by canned cycle parameter “D” (safety distance) with respect to the part.

If the depth of the groove is 0, the CNC will display the corresponding error message.

If the width of the groove is smaller than the width of the cutter (NOSEW), the CNC will display the corresponding error message.

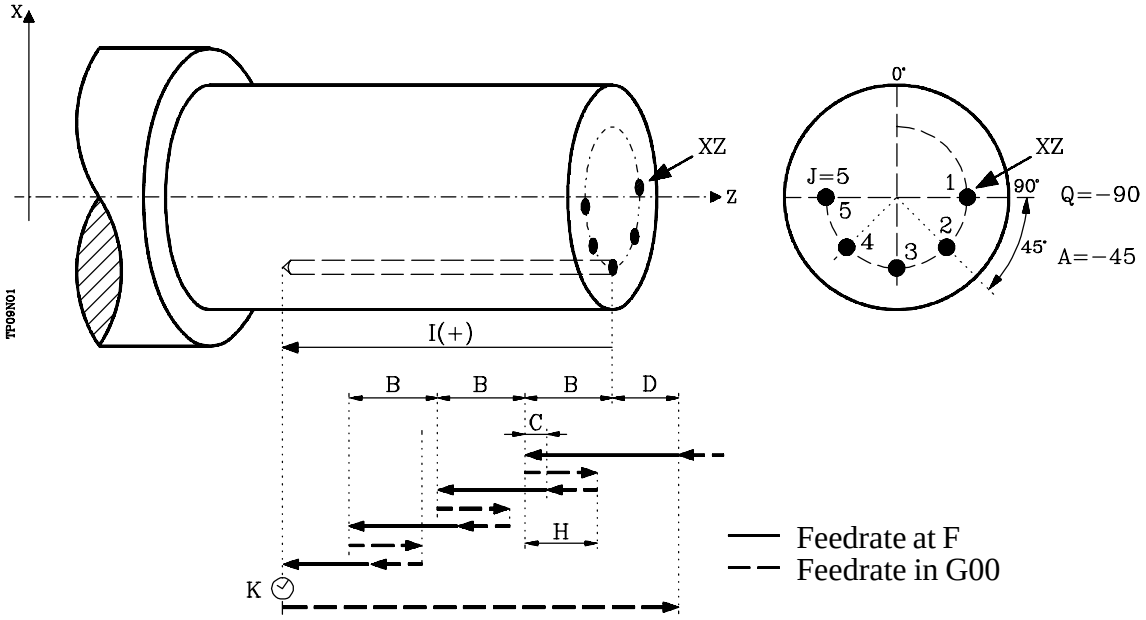
9.13 G60. AXIAL DRILLING / TAPPING

This cycle is available when the machine has a live tool.

While drilling or tapping, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.

Whether one operation or the other will be performed depends on the format being used.

For tapping, parameter B must be set to "0" and, for drilling "B" must be greater than "0.



The basic block structure for each operation is:

Drilling: **G60 X Z I B Q A J D K H C S**
Tapping: **G60 X Z I B0 Q A J D S**

X±5.5 Defines the X coordinate where the cycle will be run. It is programmed in absolute values and in the active measuring units, radius or diameter.

Z±5.5 Defines the Z coordinate where the cycle will be run. It is programmed in absolute values.

I±5.5 Defines the depth. It is referred to the starting point (X,Z). It must be assigned a positive value when drilling or tapping in the negative Z direction and negative when in doing it in the positive Z direction.

If programmed with a zero value, the CNC will issue the corresponding error.

B5.5 Defines the type of operation to be run.
* B=0 for tapping.
* B>0 for drilling, where B indicates the drilling step.

Q±5.5 Defines the first angular position to orient the spindle to drill or tap the first hole.

A±5.5 Defines the angular increment between two consecutive operations. It is programmed in degrees. Positive when counter-clockwise.

J4 Defines the number of holes to be drilled or tapped, including the first one.

If programmed with a "0" value, the CNC will issue the corresponding error.

D5.5 Defines the safety distance along the Z axis and indicates how far from the starting point (Z, X) the tool will approach the part. If not programmed, the CNC will assume "0".

K5 Defines the dwell in hundredths of a second at the bottom of the hole before starting to withdraw the tool. If not programmed, the CNC will assume "0".

The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.

H5.5 Defines the rapid (in G00) withdrawing distance along the Z axis after each drilling peck. If not programmed or programmed with a "0" value, the tool will return to the approach point.

The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.

C5.5 Defines the rapid approaching distance for successive drilling pecks. If not programmed, a distance of 1 mm (0.03937 inch) will be assumed.

The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.

S±5.5 RPM and turning direction (sign) of the live tool.

Drilling. Basic operation:

1.- Rapid move to the approach point located at a "D" distance from the first hole position.

2.- The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".

3.- Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.

4.- First drilling peck. The drilling axis moves in G01 to the programmed incremental depth "D+B".

5.- Drilling cycle. The following steps will be repeated until the bottom of the hole is reached (I).

5.1.- Rapid withdrawal (G00) a distance (H) or to the approach point.

5.2.- Rapid approach (G00) to a "C" distance from the next peck.

5.3.- New drilling peck in G01 to an incremental depth (B).

6.- Dwell "K" a the bottom of the hole in hundredths of a second, if programmed.

7.- Rapid withdrawal (G00) to the approach point.

8.- Depending on the value assigned to "J" (number of holes to be drilled).

8.1.- The spindle rotates to the new hole position. Angular increment "A".
8.2.- Repeats steps 4, 5, 6 and 7.

9.- Stops the live tool.

Tapping. Basic operation:

1.- Rapid move to the approach point located at a "D" distance from the first hole position.

2.- The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".

3.- Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.

4.- Tapping. Movement of the tapping axis to the bottom of the hole "I". MFO, SSO, FEED-HOLD and CYCLE STOP are disabled.

5.- The live tool reverses turning direction.

6.- Withdrawal (G01) to the approach point.

7.- Depending on the value assigned to "J" (number of holes to be tapped).

7.1.- The spindle rotates to the new hole position. Angular increment "A".
7.2.- Repeats steps 4, 5, 6 and 7.

8.- Stops the live tool.

Considerations

The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle.

Once the cycle has finished, the program will resume the feedrate F and the G functions which were active before calling the cycle. Only the tool radius compensation that was active will be canceled and the program will be resumed in G40.

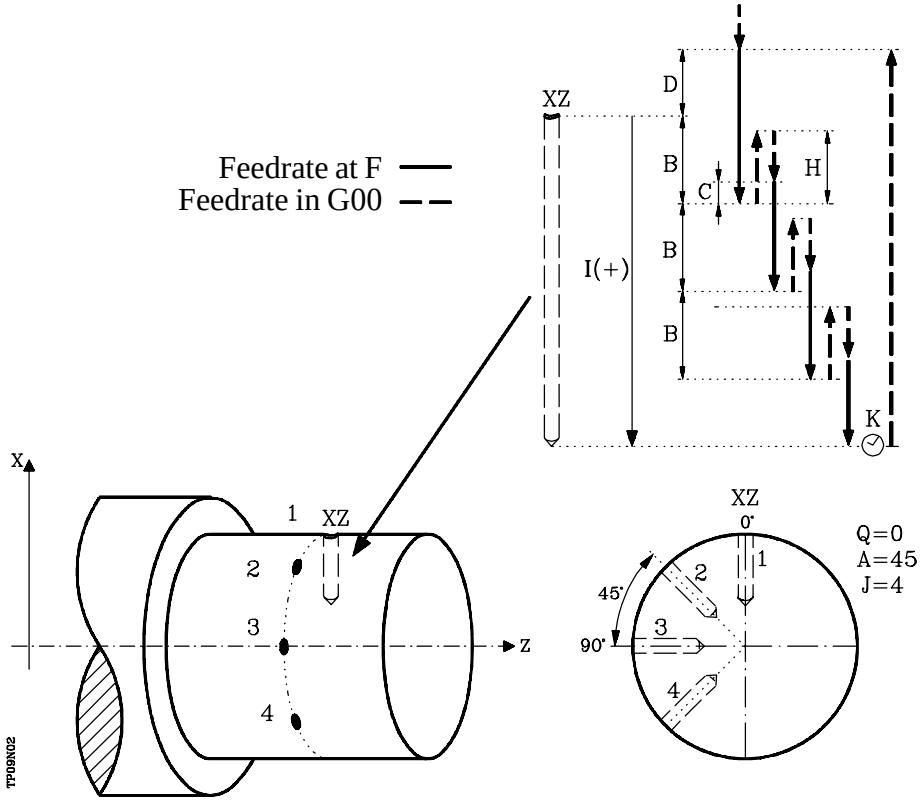
9.14 G61. RADIAL DRILLING/ TAPPING

This cycle is available when the machine has a live tool.

While drilling or tapping, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.

Whether one operation or the other will be performed depends on the format being used.

For tapping, parameter B must be set to "0" and, for drilling "B" must be greater than "0.



The basic block structure for each operation is:

Drilling: **G61 X Z I B Q A J D K H C S**
Tapping: **G61 X Z I B0 Q A J D S**

X±5.5 Defines the X coordinate where the cycle will be run. It is programmed in absolute values and in the active measuring units, radius or diameter.

Z±5.5 Defines the Z coordinate where the cycle will be run. It is programmed in absolute values.

I±5.5 Defines the depth. It is referred to the starting point (X,Z). It must be assigned a positive value when drilling or tapping in the negative X direction and negative when in doing it in the positive X direction.

If programmed with a zero value, the CNC will issue the corresponding error.

B5.5 Defines the type of operation to be run.
* B=0 for tapping.
* B>0 for drilling, where B indicates the drilling step.

Q±5.5 Defines the first angular position to orient the spindle to drill or tap the first hole.

A±5.5 Defines the angular increment between two consecutive operations. It is programmed in degrees. Positive when counter-clockwise.

J4 Defines the number of holes to be drilled or tapped, including the first one.

If programmed with a "0" value, the CNC will issue the corresponding error.

D5.5 Defines (in radius) the safety distance along the X axis and indicates how far from the starting point (Z, X) the tool will approach the part. If not programmed, the CNC will assume "0".

K5 Defines the dwell in hundredths of a second at the bottom of the hole before starting to withdraw the tool. If not programmed, the CNC will assume "0".

The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.

H5.5 Defines (in radius) the rapid (in G00) withdrawing distance along the X axis after each drilling peck. If not programmed or programmed with a "0" value, the tool will return to the approach point.

The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.

C5.5 Defines (in radius) the rapid approaching distance for successive drilling pecks. If not programmed, a distance of 1 mm (0.03937 inch) will be assumed.

The tapping operation ignores this parameter, thus not being necessary to program it. If programmed, it will be ignored.

S±5.5 RPM and turning direction (sign) of the live tool.

Drilling. Basic operation:

1.- Rapid move to the approach point located at a "D" distance from the first hole position.

2.- The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".

3.- Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.

4.- First drilling peck. The drilling axis moves in G01 to the programmed incremental depth "D+B".

5.- Drilling cycle. The following steps will be repeated until the bottom of the hole is reached (I).

5.1.- Rapid withdrawal (G00) a distance (H) or to the approach point.

5.2.- Rapid approach (G00) to a "C" distance from the next peck.

5.3.- New drilling peck in G01 to an incremental depth (B).

6.- Dwell "K" a the bottom of the hole in hundredths of a second, if programmed.

7.- Rapid withdrawal (G00) to the approach point.

8.- Depending on the value assigned to "J" (number of holes to be drilled).

8.1.- The spindle rotates to the new hole position. Angular increment "A".
8.2.- Repeats steps 4, 5, 6 and 7.

9.- Stops the live tool.

Tapping. Basic operation:

1.- Rapid move to the approach point located at a "D" distance from the first hole position.

2.- The CNC turns on the live tool at the rpm and in the turning direction indicated by parameter "S".

3.- Orients the spindle at the angular position set by "Q". If the spindle was turning, it will stop.

4.- Tapping. Movement of the tapping axis to the bottom of the hole "I". MFO, SSO, FEED-HOLD and CYCLE STOP are disabled.

5.- The live tool reverses turning direction.

6.- Withdrawal (G01) to the approach point.

7.- Depending on the value assigned to "J" (number of holes to be tapped).

7.1.- The spindle rotates to the new hole position. Angular increment "A".
7.2.- Repeats steps 4, 5, 6 and 7.

8.- Stops the live tool.

Considerations

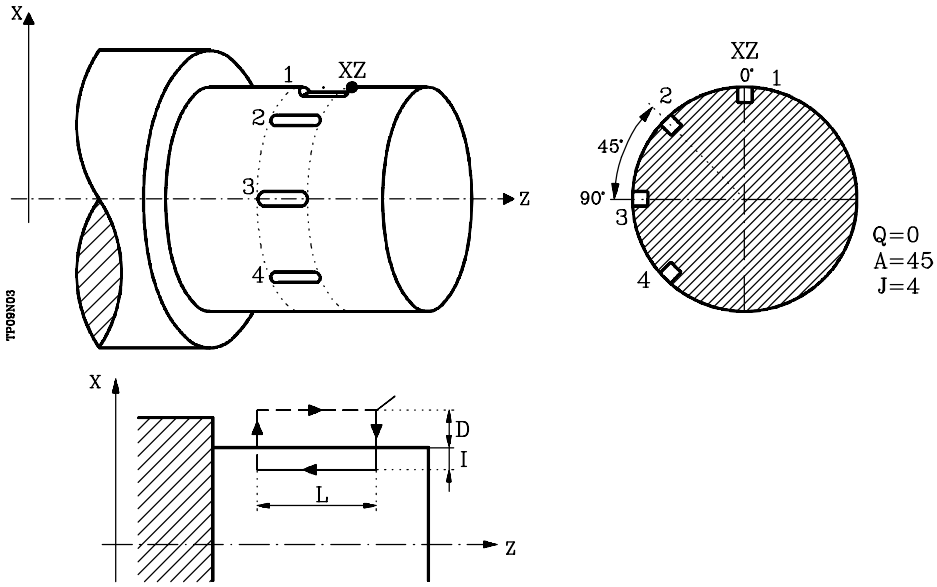
The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle.

Once the cycle has finished, the program will resume the feedrate F and the G functions which were active before calling the cycle. Only the tool radius compensation that was active will be canceled and the program will be resumed in G40.

9.15 G62. LONGITUDINAL SLOT MILLING CYCLE

This cycle is available when the machine has a live tool.

While performing this operation, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.



The basic block structure is: **G62 X Z L I Q A J D F S**

X±5.5 Defines the X coordinate where the cycle is to be run. It is programmed in absolute values and in the active units, radius or diameter.

Z±5.5 Defines the Z coordinate where the cycle is to be run. It is programmed in absolute coordinates.

L±5.5 Defines the length of the slot. It is referred to the starting point (X, Z) and it will, therefore, have a positive value when machining in the negative Z direction and negatitve when doing it in the opposite direction. "L(+)" in the example above.

If programmed with a "0" value, the CNC will issue the corresponding error.

I±5.5 Defines, in radius, the depth of the slot. It is referred to the starting point (X,Z).

If programmed with a "0" value, the CNC will issue the corresponding error.

Q±5.5 Defines the angular position (in degrees) to orient the spindle to mill the first slot.

A±5.5 Defines the angular increment between two consecutive slots. It is programmed in degrees with the positive sign when going counter-clockwise.

J4 Defines the number of slots to be milled (first one included). If "0" is programmed, the CNC will issue the corresponding error message.

D5.5 Defines, in radius, the safety distance along the X axis and it indicates how far from the starting point (X, Z) the tool is to approach the part. If not programmed, the CNC will assume a value of "0".

F5.5 Defines the slot milling feedrate.

S±5.5 RPM and turning direction (sign) of the live tool.

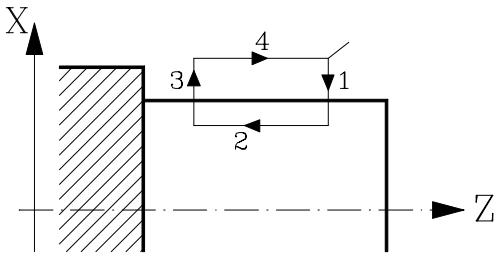
Basic operation:

1.- Rapid approach upto the safety distance "D" from the slot milling point.

2.- The CNC turns the live tool on at the rpm and turning direction set by parameter "S".

3.- Orients the spindle to the indicated angular position "Q", If the spindle was turning, the CNC will stop it.

4.- Milling of the slot by following these steps:



4.1.- In feed at the feedrate selected prior to calling the cycle.
4.2.- Milling of the slot by moving the Z axis at the programmed "F" feedrate.
4.3.- Rapid withdrawal to the reference point.
4.4.- Rapid return to the starting point.

5.- Depending on the setting of parmeter "J" (number of slots):

5.1.- The spindle orients to the new position. Angular increment "A".
5.2.- Repeats the movements indicated on point 4.

6.- The live tool stops.

Considerations

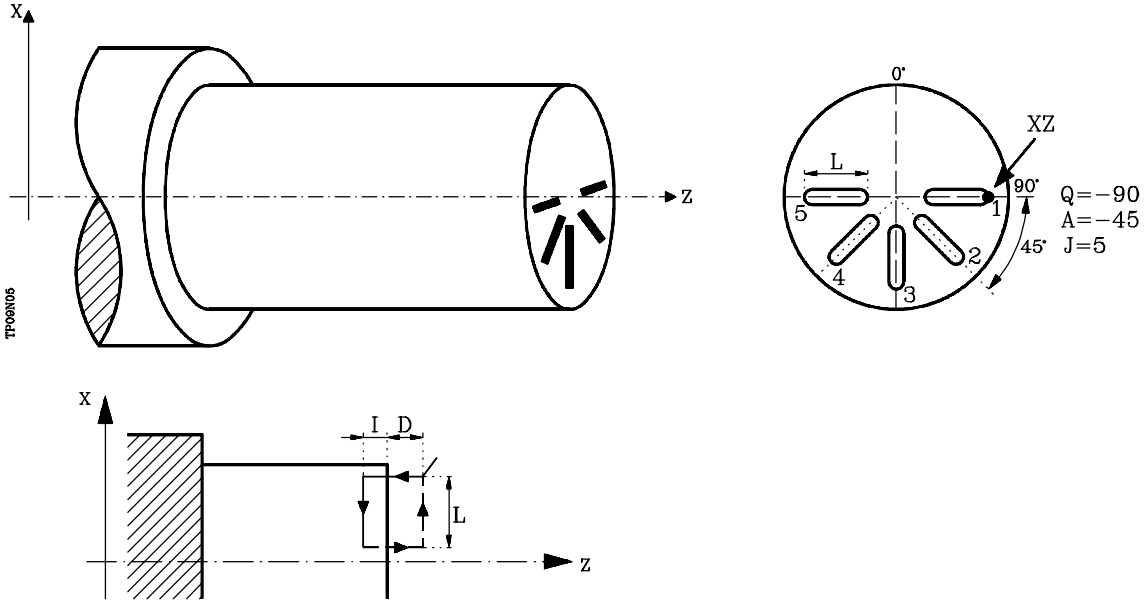
The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle.

Once the cycle has finished, the program will resume the feedrate F and the G functions which were active before calling the cycle. Only the tool radius compensation that was active will be canceled and the program will be resumed in G40.

9.16 G63. RADIAL SLOT MILLING CYCLE

This cycle is available when the machine has a live tool.

While performing this operation, the spindle is stopped and the tool is turning. This makes it possible to machine anywhere on the workpiece.



The basic block structure is: **G63 X Z L I Q A J D F S**

X±5.5 Defines the X coordinate where the cycle is to be run. It is programmed in absolute values and in the active units, radius or diameter.

Z±5.5 Defines the Z coordinate where the cycle is to be run. It is programmed in absolute coordinates.

L±5.5 Defines, in radius, the length of the slot. It is referred to the starting point (X, Z) and it will, therefore, have a positive value when machining in the negative X direction and negatitve when doing it in the opposite direction. "L(+)" in the example above.

If programmed with a "0" value, the CNC will issue the corresponding error.

I±5.5 Defines the depth of the slot. It is referred to the starting point (X,Z).

If programmed with a "0" value, the CNC will issue the corresponding error.

Q±5.5 Defines the angular position (in degrees) to orient the spindle to mill the first slot.

A±5.5 Defines the angular increment between two consecutive slots. It is programmed in degrees with the positive sign when going counter-clockwise.

J4 Defines the number of slots to be milled. If "0" is programmed, the CNC will issue the corresponding error message.

D5.5 Defines the safety distance along the Z axis and it indicates how far from the starting point (X, Z) the tool is to approach the part. If not programmed, the CNC will assume a value of "0".

F5.5 Defines the slot milling feedrate.

S±5.5 RPM and turning direction (sign) of the live tool.

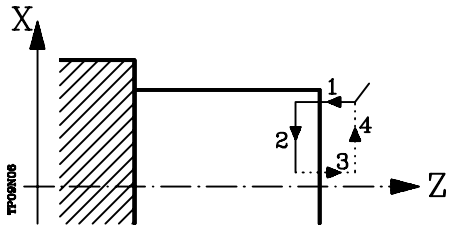
Basic operation:

1.- Rapid approach upto the safety distance "D" from the slot milling point.

2.- The CNC turns the live tool on at the rpm and turning direction set by parameter "S".

3.- Orients the spindle to the indicated angular position "Q", If the spindle was turning, the CNC will stop it.

4.- Milling of the slot by following these steps:



4.1.- In feed at the feedrate selected prior to calling the cycle.
4.2.- Milling of the slot by moving the X axis at the programmed "F" feedrate.
4.3.- Rapid withdrawal to the reference point.
4.4.- Rapid return to the starting point.

5.- Depending on the setting of parmeter "J" (number of slots):

5.1.- The spindle orients to the new position. Angular increment "A".
5.2.- Repeats the movements indicated on point 4.

6.- The live tool stops.

Considerations

The machining conditions (feedrate, live tool rpm, etc.) must be programmed prior to the calling the cycle.

Once the cycle has finished, the program will resume the feedrate F and the G functions which were active before calling the cycle. Only the tool radius compensation that was active will be canceled and the program will be resumed in G40.

10. PROBING

The CNC has two probe inputs, one for 5V TTL signals and the other for 24V.

The appendix to the Installation and Start-up manual explains the connections of the various probe types to these inputs.

This CNC allows, by using probes, the following operations to be performed:

* Programming probing movements by means of function G75.

* Executing the various tool calibration and part measurement cycles by programming high level blocks.

10.1 PROBING (G75,G76)

The **G75** function allows movements to be programmed which will end after the CNC **receives** the signal from the measuring probe used.

The **G76** function allows movements to be programmed which will end after the CNC **no longer receives** the signal from the measuring probe used.

Their definition format is:

G75 X..C ± 5.5
G76 X..C ± 5.5

After **G75** or **G76**, the required axis or axes will be programmed, as well as the coordinates of these axes which will define the end point of the programmed movement.

The machine will move according to the programmed path until it receives the signal from the probe (G75) or until it no longer receives the probe signal (G76). At this time, the CNC will consider the block finished, taking as the theoretical position of the axes the real position which they have at that time.

If the axes reach the programmed position before receiving (G75) or while receiving (G76) the external signal from the probe, the CNC will stop the movement of the axes.

This type of movement with probing blocks are very useful when it is required to generate measurement or verification programs for tools and parts.

Functions G75 and G76 are not modal and, therefore, must be programmed whenever it is wished to probe.

It is not possible to vary the Feedrate Override while either G75 or G76 is active. It stays set at 100 %.

Functions G75 and G76 are incompatible with each other and with G00, G02, G03, G33, G41 and G42 functions. In addition, once this has been performed, the CNC will assume functions G01 and G40.

10.2 PROBING CANNED CYCLES

The CNC offers the following probing canned cycles:

1 Tool calibration canned cycle.

2 Probe calibration canned cycle.

3 Part measuring and tool offset correction canned cycle along X axis.

4 Part measuring and tool offset correction canned cycle along Z axis.

All movements of these canned cycles will be executed along the X and Z axes.

The canned cycles will be programmed by means of the high level statement PROBE being its format:

(PROBE (expression), (assignment statement), ...)

This statement calls the probing canned cycle indicated by a number or by an expression which results in a number. The parameters of such cycle can also be initialized with the desired values contained in the assignment statement.

General considerations

The probing canned cycles are not modal, therefore, they must be programmed every time it is desired to execute them.

The probes used to execute these cycles are:

* Probe, placed at a set position of the machine, used to calibrate tools.

* Probe, placed at the spindle, that will be treated like a tool and will be used in the various measuring canned cycles.

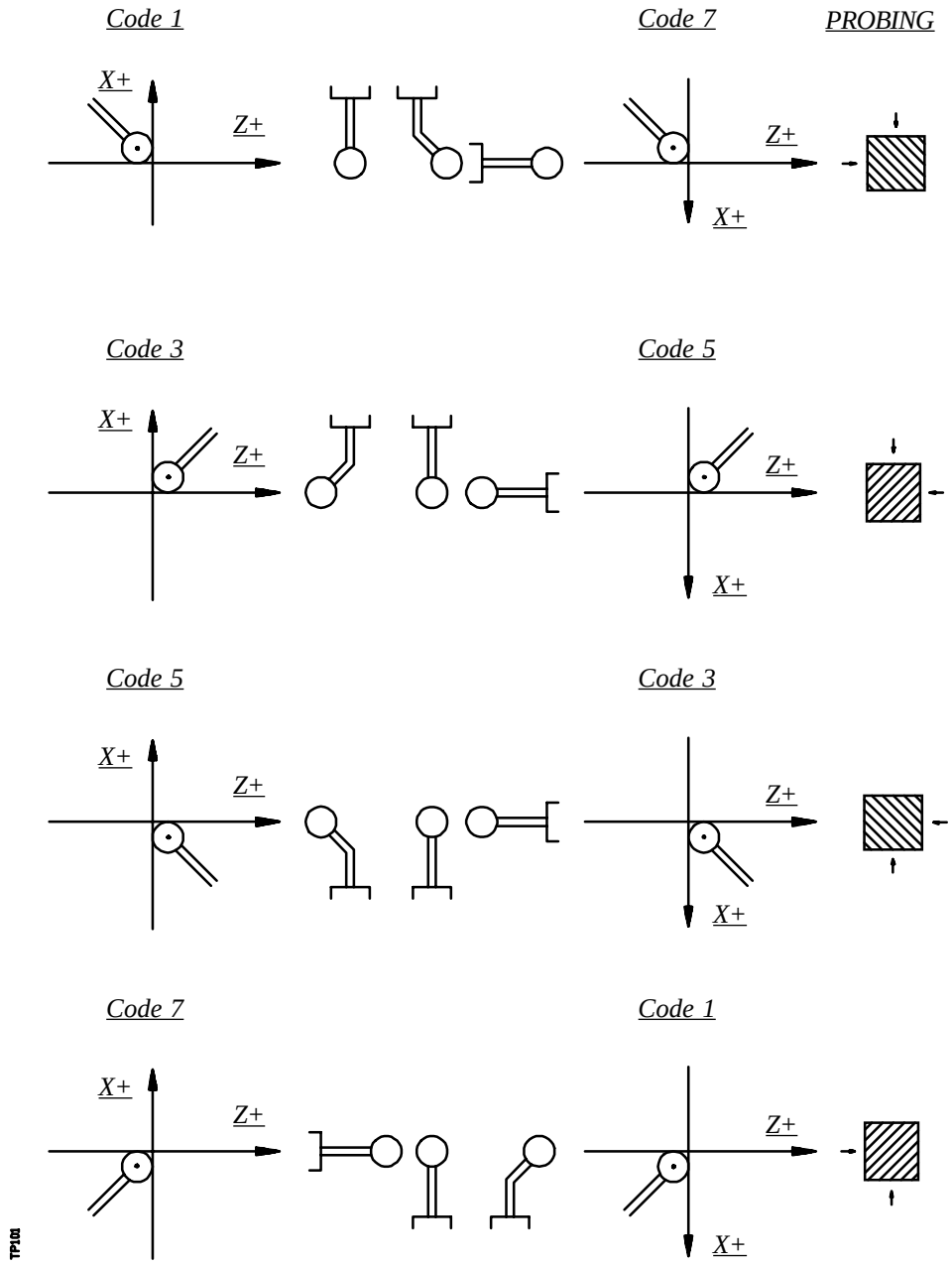
The execution of a probing canned cycle does not alter the history of previous G functions except those concerning tool radius compensation G41 and G42.

10.3 TOOL CALIBRATION CANNED CYCLE

Used to calibrate a tool or a probe placed at the spindle (tool holder). Once completed the cycle, the X and Z values of their tool offset table are updated and their I and K values will be reset to zero.

When calibrating the tool or the probe for the first time, it is recommended to enter in their tool offset table approximate values of their lengths (X, Z) as well as their location codes (F) and radius (R).

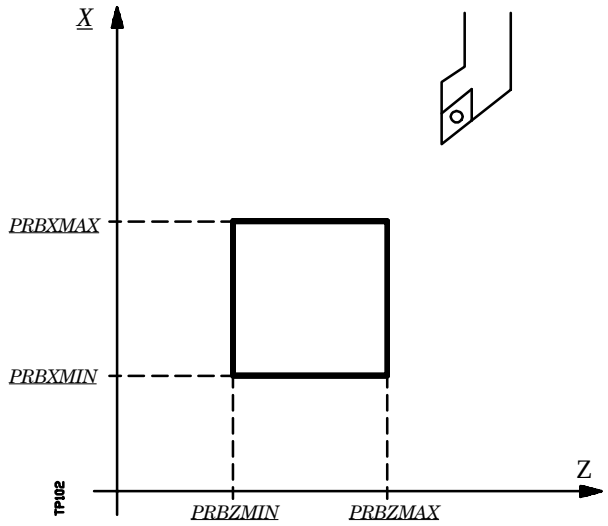
For a probe, the "R" value will be the value of the radius of the stylus (ball) and its location code will depend on how the calibration is carried out.



To execute this cycle, it is required to use a table-top probe placed in a set position on the machine and with its sides parallel to the X and Z axes.

Its position will be indicated in absolute coordinates referred to Machine Reference Zero by means of the following general machine parameters:

PRBXMIN	indicates the minimum X coordinate of the probe.
PRBXMAX	indicates the maximum X coordinate of the probe.
PRBZMIN	indicates the minimum Z coordinate of the probe.
PRBZMAX	indicates the maximum Z coordinate of the probe.



The programming format for this cycle is:

(PROBE 1, B, F, X, U, Z, W)

B5.5 Defines the safety distance with a positive radius value.

If programmed with a value of 0, the CNC will display its corresponding error message.

F5.5 Defines the probing feedrate in mm/min. or inches/min.

X, U, Z, W are optional parameters which usually need not be defined.

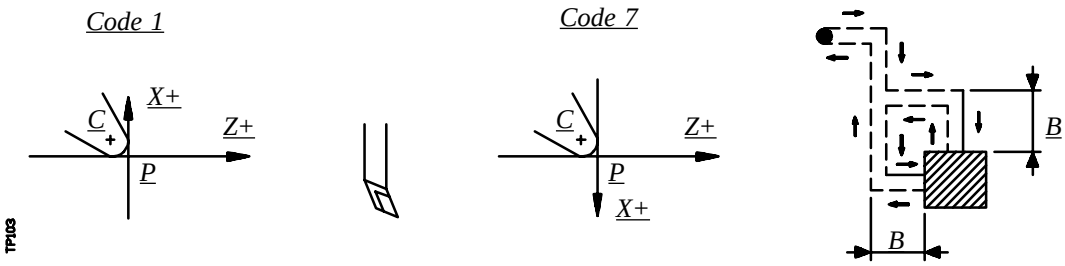
On some machines, due to the lack of probe positioning repeatability, the probe must be calibrated before each calibration.

Instead of redefining machine parameters PRBXMIN, PRBXMAX, PRBZMAX, and PRBZMIN every time the probe is calibrated, these coordinates may be indicated in the X, U, Z and W variables respectively.

The CNC does not change the machine parameters but considers the coordinates indicated in X, U, Z, W only during this calibration.

If any of the X, U, Z, W fields is left out, the CNC assumes the value assigned to the corresponding machine parameter.

Basic operation:



1.- Approach move

Rapid move (G00) from the cycle calling point to the approach corner.

This point is located in front of the corresponding probe's corner at a safety distance (B) from both sides.

The approach is made in two stages:

1st Move along Z axis.

2nd Move along X axis.

2.- Probing move

Depending on the location code assigned to the selected tool, one or two probings will be carried out for its calibration. Each probing operation will follow these steps:

a.- Approach move

Rapid move (G00) up to the approach point located in front of the side to be probed and at a distance "B" from it.

b.- Probing move

Probe movement at indicated feedrate (F) until probe signal is received.

The maximum distance to travel in each probing move is 2B. If after travelling this distance the CNC has not yet received the probe signal, it will display the corresponding error message and it will stop the axes.

c.- Withdrawal

Rapid move of the probe (G00) from the last touched point back to the approach corner.

3.- Withdrawal

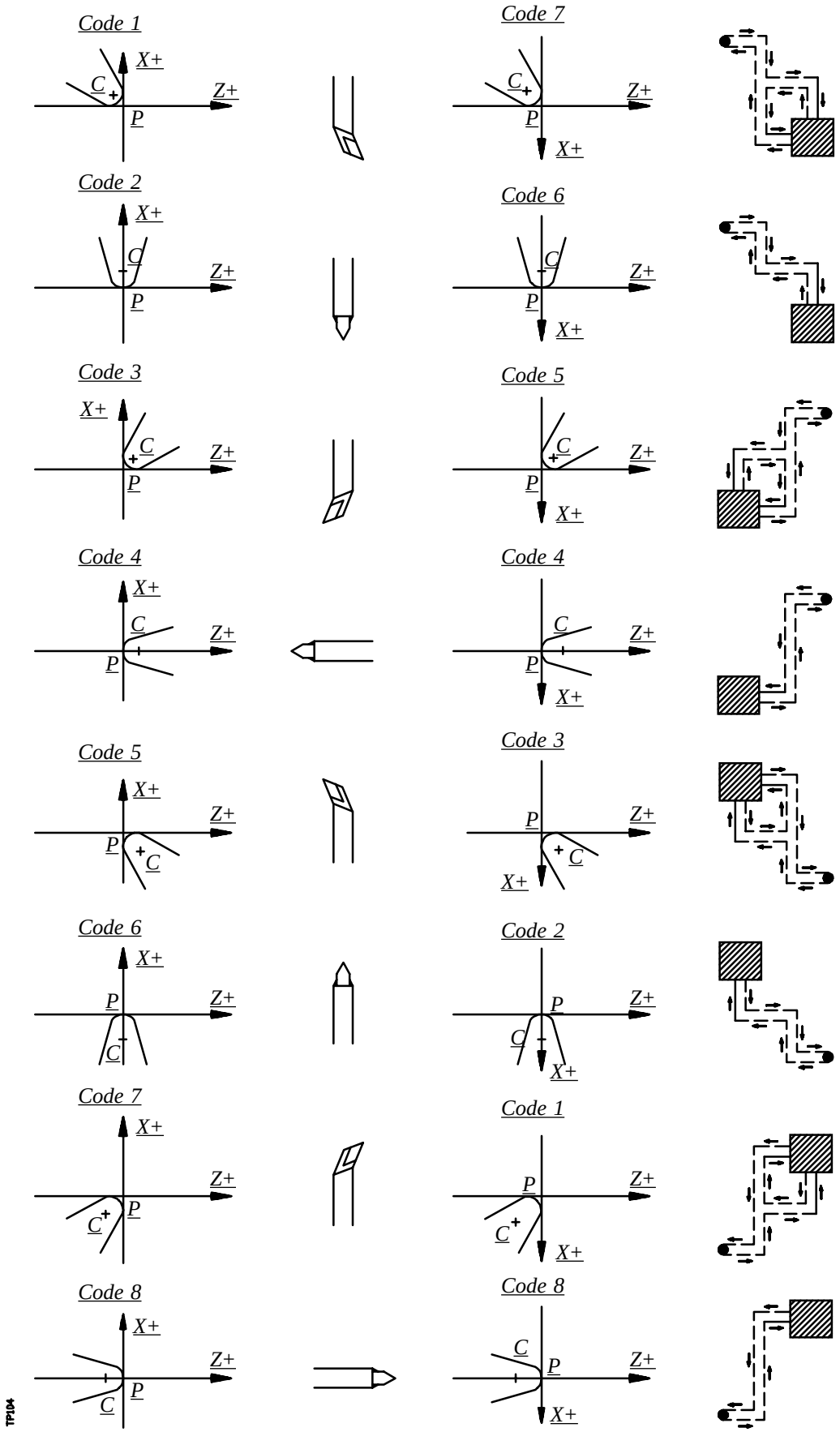
Rapid move (G00) from the approach corner back to the cycle calling point.

The withdrawal will be carried out in two stages:

1st Movement along X axis.

2nd Movement along Z axis.

Once the cycle has ended, the CNC will have updated the X and Z values of tool offset table corresponding to the calibrated tool as well as reset its I and K values to zero.


Code 1
X+
C
Z+
P
Code 7
C
Z+
P
X+
Code 2
X+
C
Z+
P
Code 6
C
Z+
P
X+
Code 3
X+
C
Z+
P
Code 5
C
Z+
P
X+
Code 4
X+
C
Z+
P
Code 4
C
Z+
P
X+
Code 5
X+
Z+
P
C
Code 3
P
Z+
C
X+
Code 6
X+
P
Z+
C
Code 2
P
Z+
C
X+
Code 7
X+
Z+
C
P
Code 1
P
Z+
C
X+
Code 8
X+
Z+
C
P
Code 8
C
P
Z+
X+
TP104

Also, the CNC will return the detected errors in the following global arithmetic parameters:

P298 **Error detected along X axis.** Difference between the real tool length and the value assigned as X tool length in the corresponding tool offset table. This value will be given in radius.

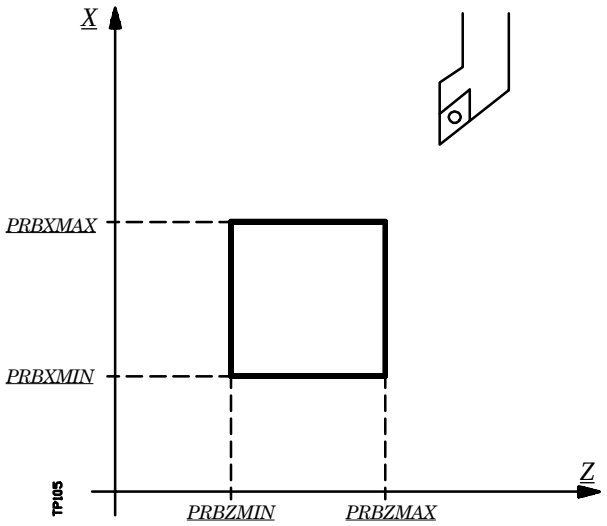
P299 **Error detected along Z axis.** Difference between the real tool length and the value assigned as Z tool length in the corresponding tool offset table.

10.4 PROBE CALIBRATION CANNED CYCLE

Used to calibrate the sides of the table-top probe, installed in a set position on the machine and with its sides parallel to the X and Z axes.

This probe will be used later to calibrate tools. Its approximate position must be indicated in the following general machine parameters in absolute coordinate values referred to Machine Reference Zero:

PRBXMIN	indicates the minimum X coordinate of the probe.
PRBXMAX	indicates the maximum X coordinate of the probe.
PRBZMIN	indicates the minimum Z coordinate of the probe.
PRBZMAX	indicates the maximum Z coordinate of the probe.



To execute this cycle, it is required to use a reference tool of known dimensions whose corresponding values have already been entered in the tool offset table.

Since the probe must be calibrated along both X and Z axis, the location code (F) of the selected reference tool must be F1, F3, F5 or F7.

The programming format for this cycle is:

(PROBE 2, B, F, X, U, Z, W)

B5.5 Defines the safety distance with a positive radius value.

If programmed with a value of 0, the CNC will display its corresponding error message.

F5.5 Defines the probing feedrate in mm/min. or inches/min.

X, U, Z, W are optional parameters which usually need not be defined.

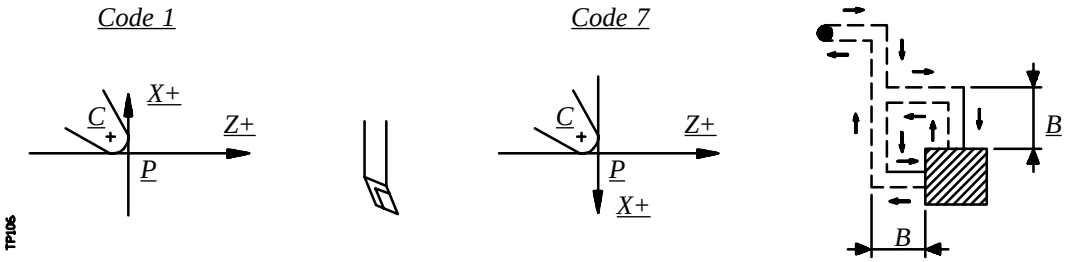
On some machines, due to the lack of probe positioning repeatability, the probe must be calibrated before each calibration.

Instead of redefining machine parameters PRBXMIN, PRBXMAX, PRBZMAX, and PRBZMIN every time the probe is calibrated, these coordinates may be indicated in the X, U, Z and W variables respectively.

The CNC does not change the machine parameters but considers the coordinates indicated in X, U, Z, W only during this calibration.

If any of the X, U, Z, W fields is left out, the CNC assumes the value assigned to the corresponding machine parameter.

Basic operation:



1.- Approach move

Rapid move (G00) from the cycle calling point to the approach corner.

This point is located in front of the corresponding probe's corner at a safety distance (B) from both sides.

The approach is made in two stages:

1st Move along Z axis.

2nd Move along X axis.

2.- Probing move

The probe sides used in this probing move as well as the tool path depEnd on the location code assigned to the selected tool.

This stage has two probing movements and each one of them consists of the following steps:

a.- Approach move

Rapid move (G00) up to the approach point located in front of the side to be probed and at a distance "B" from it.

b.- Probing move

Probe movement at indicated feedrate (F) until probe signal is received.

The maximum distance to travel in each probing move is 2B. If after travelling this distance the CNC has not yet received the probe signal, it will display the corresponding error message and it will stop the axes.

c.- Withdrawal

Rapid move (G00) from the last touched point back to the approach corner.

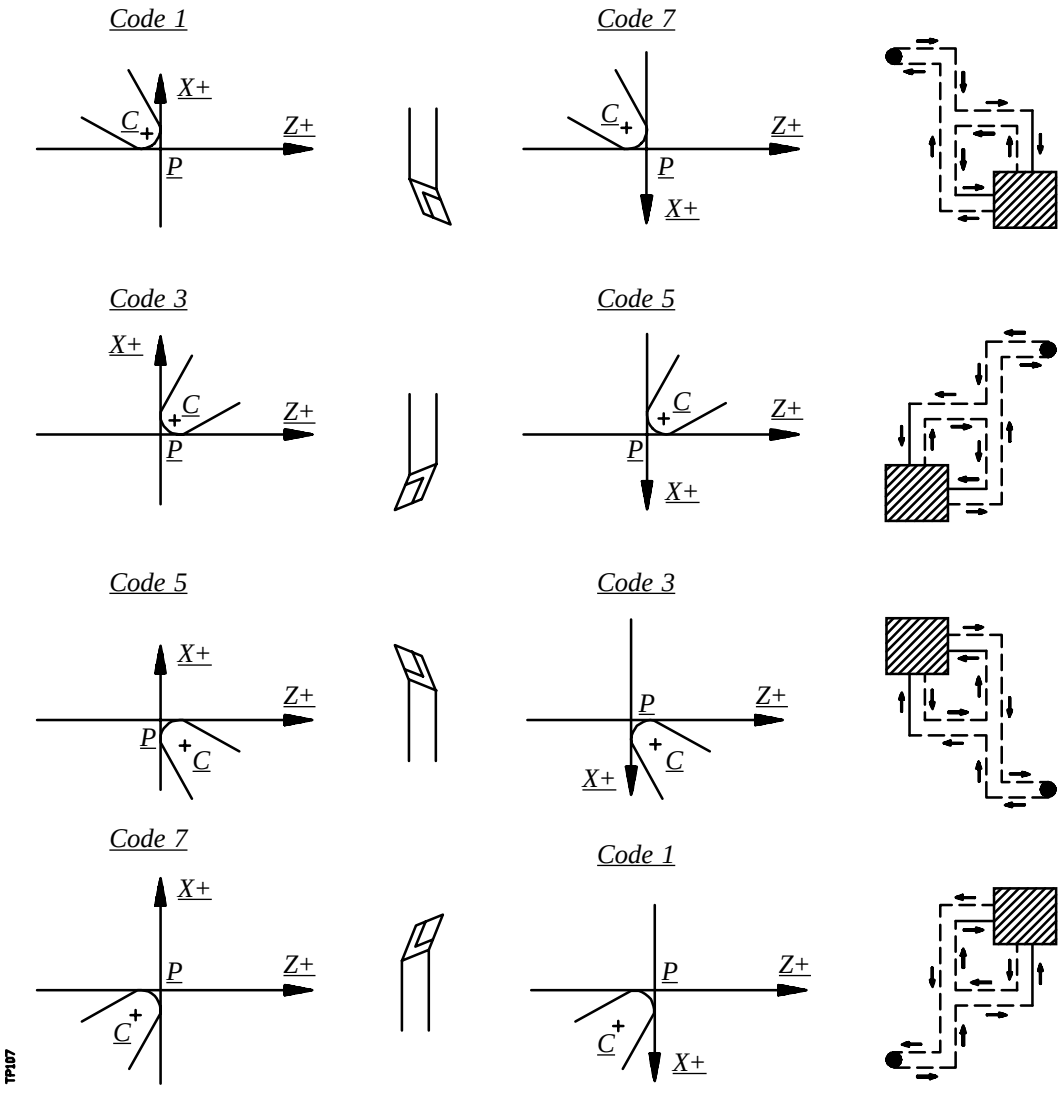
3.- Withdrawal

Rapid move (G00) from the approach corner back to the cycle calling point.

The withdrawal will be carried out in two stages:

1st Movement along X axis.

2nd Movement along Z axis.



Once the cycle has ended, the CNC will return the measured values in the following global arithmetic parameters:

P298 **Real X coordinate of the measured side.** Given in absolute with respect to Machine Reference Zero and in radius.

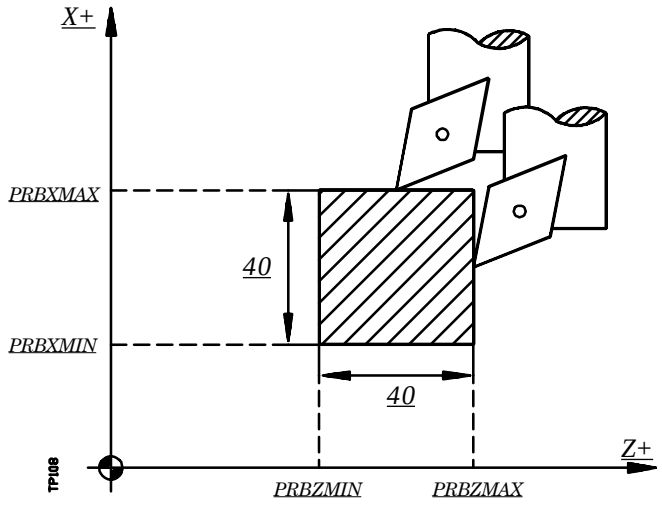
P299 **Real Z coordinate of the measured side.** Given in absolute with respect to Machine Reference Zero.

Knowing these values and the probe dimensions, the user will update the following general machine parameters after calculating the position values for the other sides of the probe.

PRBXMIN	indicates the minimum X coordinate of the probe.
PRBXMAX	indicates the maximum X coordinate of the probe.
PRBZMIN	indicates the minimum Z coordinate of the probe.
PRBZMAX	indicates the maximum Z coordinate of the probe.

Example:

If the tool used has a location code of F3 and the probe has a square stylus with a side of 40mm, the values to be assigned to these parameters will be:



PRBXMIN = P298 - 40 mm
PRBXMAX = P298
PRBZMIN = P299 - 40 mm
PRBZMAX = P299

10.5 PART MEASURING AND TOOL OFFSET CORRECTION CANNED CYCLE ALONG X AXIS

A previously calibrated probe (with the cycle described before) placed in the spindle (tool holder) will be used.

With this cycle, besides being possible to measure the part along the X axis, it is also possible to correct the offset of the tool used to machine this surface. This correction will only take place when the measured error is greater than a pre-established value.

The programming format for this cycle is:

(PROBE 3, X, Z, B, F, L, D)

X±5.5 Theoretical X coordinate of the point to be measured. This value will be given in the active units (radius or diameter).

Z±5.5 Theoretical Z coordinate of the point to be measured.

B5.5 Defines the safety distance with a positive radius value.

If a value of 0 is programmed, the CNC will display the corresponding error message.

F5.5 Defines the probing feedrate in mm/min. or inches/min.

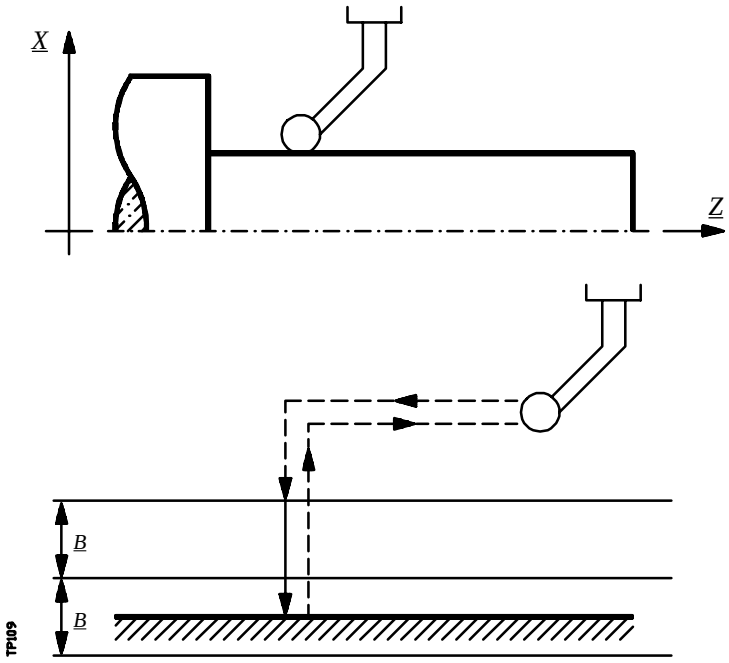
L5.5 Defines the tolerance amount to be applied to the measured error. It will be programmed in radius and the correction will only take place if the error is equal to or greater than this value.

If not programmed, the CNC will assume a value of 0 for this parameter.

D4 Defines the number of the tool offset being corrected once the measuring cycle has ended.

If not programmed or programmed with a value of 0, the CNC will assume that the tool offset must not be corrected.

Basic operation:



1.- Approach move

Rapid move of the probe (G00) from the cycle calling point to the approach point.

This point is located in front of the point to be measured and at a safety distance (B) from it.

The approach movement is carried out in two stages:

1st Move along Z axis.

2nd Move along X axis.

2.- Probing

Movement of the probe along the X axis at the indicated feedrate (F) until the probe signal is received.

The maximum distance to travel in this move is 2B. If after travelling this distance, the CNC has not yet received the probe signal, it will issue the corresponding error message and it will stop the axes.

Once the probing has ended, the CNC will assume as theoretical position the real position at the time when it received the probe signal.

3.- Withdrawal

Rapid movement of the probe (G00) from the probed point to the cycle calling point.

The withdrawal is done in two stages:

1st X axis move back to the cycle calling point.

2nd Z axis move back to the cycle calling point.

Once the cycle has ended, the CNC will return the real values, obtained from the measurements, in the global arithmetic parameters:

P298 Real X value of the measured point in active units (radius or diameter).

P299 **Detected error.** Difference between the real coordinate and the programmed theoretical one. This value is given in radius.

If a tool offset number (D) was selected, the CNC will alter its "I" value as long as the measured error is equal to or greater than the allowed tolerance "L".

10.6 PART MEASURING AND TOOL OFFSET CORRECTION CANNED CYCLE ALONG Z AXIS

A previously calibrated probe (with the cycle described before) placed in the spindle (tool holder) will be used.

With this cycle, besides being possible to measure the part along the Z axis, it is also possible to correct the offset of the tool used to machine this surface. This correction will only take place when the measured error is greater than a pre-established value.

The programming format for this cycle is:

(PROBE 4, X, Z, B, F, L, D)

X±5.5 Theoretical X coordinate of the point to be measured. This value will be given in the active units (radius or diameter).

Z±5.5 Theoretical Z coordinate of the point to be measured.

B5.5 Defines the safety distance with a positive radius value.

If a value of 0 is programmed, the CNC will display the corresponding error message.

F5.5 Defines the probing feedrate in mm/min. or inches/min.

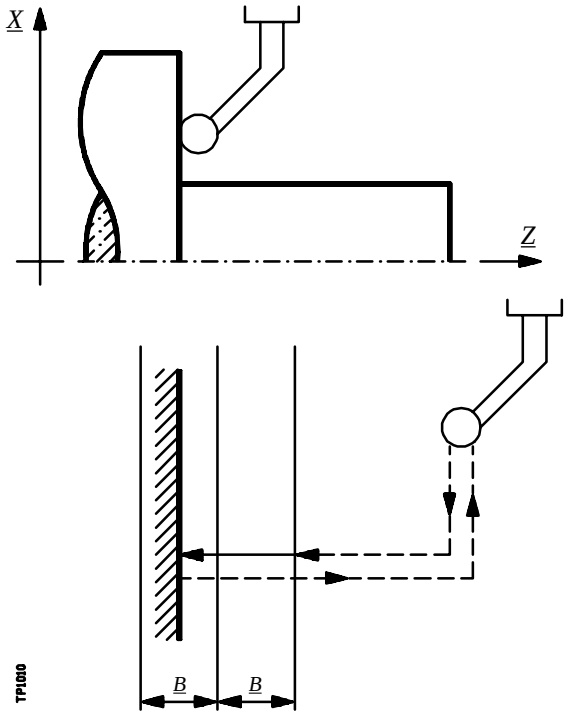
L5.5 Defines the tolerance amount to be applied to the measured error. It will be programmed in radius and the correction will only take place if the error is equal to or greater than this value.

If not programmed, the CNC will assume a value of 0 for this parameter.

D4 Defines the number of the tool offset being corrected once the measuring cycle has ended.

If not programmed or programmed with a value of 0, the CNC will assume that the tool offset must not be corrected.

Basic operation:



1.- Approach move

Rapid move of the probe (G00) from the cycle calling point to the approach point.

This point is located in front of the point to be measured and at a safety distance (B) from it.

The approach movement is carried out in two stages:

1st Move along Z axis.

2nd Move along X axis.

2.- Probing

Movement of the probe along the Z axis at the indicated feedrate (F) until the probe signal is received.

The maximum distance to travel in this move is 2B. If after travelling this distance, the CNC has not yet received the probe signal, it will issue the corresponding error message and it will stop the axes.

Once the probing has ended, the CNC will assume as theoretical position the real position at the time when it received the probe signal.

3.- Withdrawal

Rapid movement of the probe (G00) from the probed point to the cycle calling point.

The withdrawal is done in two stages:

1st Z axis move back to the cycle calling point.

2nd X axis move back to the cycle calling point.

Once the cycle has ended, the CNC will return the real values, obtained from the measurements, in the global arithmetic parameters:

P298 Real Z value of the measured point.

P299 Detected error. Difference between the real coordinate and the programmed theoretical one.

If a tool offset number (D) was selected, the CNC will alter its "K" value as long as the measured error is equal to or greater than the allowed tolerance "L".

11. PROGRAMMING IN HIGH-LEVEL LANGUAGE

The CNC has a series of internal variables which can be accessed from the user program, from the PLC program or through DNC.

Access to these variables from the user program is gained with high-level commands.

Each of the system variables which can be accessed will be referred to by means of its mnemonic, and will be separated, according to their use, into read-only variables and read-write variables.

11.1 LEXICAL DESCRIPTION

All the words which form the high-level language of the numerical control must be written in capital letters except for associated texts which may be written in upper and lower case letters.

The following elements are available for high-level programming:

- Reserved words.
- Numerical constants
- Symbols

11.1.1 RESERVED WORDS

The set of words which the CNC uses in high-level programming for naming system variables, operators, control mnemonics, etc. are as follows:

ANAIn	**ANAOn**	**BLKN**	**CALL**	**CALLP**	**CLOCK**
CNCERR	**CNCFRO**	**CNCSSO**	**CYTIME**	**DATE**	**DEFLEX**
DEFLEY	**DEFLEZ**	**DFHOLD**	**DIGIT**	**DIST(X-C)**	**DNCERR**
DNCF	**DNCFPR**	**DNCFRO**	**DNCS**	**DNCSL**	**DNCSSO**
DPOS(X-C)	**DSBLK**	**DSTOP**	**DW**	**EFHOLD**	**ERROR**
ESBLK	**ESTOP**	**EXEC**	**FEED**	**FIRST**	**FLWE(X-C)**
FLWES	**FOZLO(X-C)**	**FOZONE**	**FOZUP(X-C)**	**FPREV**	**FRO**
FZLO(X-C)	**FZONE**	**FZUP(X-C)**	**GGSA**	**GGSB**	**GGSC**
GGSD	**GMS**	**GOTO**	**GSn**	**GTRATY**	**GUP n**
IB	**IF**	**INPUT**	**KEY**	**KEYSRC**	**LONGAX**
LUP (a,b)	**MCALL**	**MDOFF**	**MIRROR**	**MP(X-C)n**	**MPASn**
MPGn	**MPLCn**	**MPSn**	**MPSSn**	**MSG**	**MSn**
NBTOOL	**NXTOD**	**NXTOOL**	**ODW**	**OPEN**	**OPMODA**
OPMODB	**OPMODC**	**OPMODE**	**ORG(X-C)**	**ORG(X-C)n**	**ORGROA**
ORGROB	**ORGROC**	**ORGROI**	**ORGROJ**	**ORGROK**	**ORGROQ**
ORGROR	**ORGROS**	**ORGROT**	**ORGROX**	**ORGROY**	**ORGROZ**
PAGE	**PARTC**	**PCALL**	**PLANE**	**PLCCn**	**PLCERR**
PLCF	**PLCFPR**	**PLCFRO**	**PLCIn**	**PLCMn**	**PLCMSG**
PLCOF(X-C)	**PLCOn**	**PLCRn**	**PLCS**	**PLCSL**	**PLCSSO**
PLCTn	**PORGF**	**PORGS**	**POS(X-C)**	**POSS**	**PPOS(X-C)**
PRBST	**PRGF**	**PRGFIN**	**PRGFPR**	**PRGFRO**	**PRGN**
PRGS	**PRGSL**	**PRGSSO**	**PROBE**	**REPOS**	**RET**
ROTPF	**ROTPS**	**RPOSS**	**RPT**	**RTPOSS**	**SCALE**
SCALE(X-C)	**SCNCSO**	**SDNCS**	**SDNCSL**	**SDNCSO**	**SFLWES**
SK	**SLIMIT**	**SPEED**	**SPLCS**	**SPLCSL**	**SPLCSO**
SPOSS	**SPRGS**	**SPRGSL**	**SPRGSO**	**SREAL**	**SRPOSS**
SRTPOS	**SSLIMI**	**SSO**	**SSPEED**	**SSREAL**	**SSSO**
STPOSS	**SUB**	**SYMBOL**	**SYSTEM**	**SZLO(X-C)**	**SZONE**
SZUP(X-C)	**TIME**	**TIMER**	**TLFDn**	**TLFFn**	**TLFNn**
TLFRn	**TMZPn**	**TMZTn**	**TOD**	**TOIn**	**TOKn**
TOLn	**TOOL**	**TOOROF**	**TOOROS**	**TORn**	**TPOS(X-C)**
TPOSS	**TRACE**	**TZLO(X-C)**	**TZONE**	**TZUP(X-C)**	**WBUF**
WBUF	**WKEY**	**WRITE**			

Words ending in (X-C) indicate a set of 9 elements formed by the corresponding root followed by X,Y,Z,U,V,W,A,B and C.

ORG(X-C) --> ORGX,ORGY,ORGZ,ORGU,ORGV,ORGW,ORGA,ORGB, ORGC

All the letters of the alphabet A-Z are also reserved words, as they can make up a high-level language word when used alone.

11.1.2 NUMERICAL CONSTANTS

The blocks programmed in high-level language allow numbers in decimal format which do not exceed the format #6.5 and numbers in hexadecimal format, in which case they must be preceded by the **$** sign, with a maximum of 8 digits.

The assignment to a variable of a constant higher than the format #6,5 will be made by means of arithmetic parameters, by means of arithmetic expressions or by means of constants expressed in hexadecimal format.

Example: To assign the value 100000000 to the variable "TIMER", It can be done in one of the following ways:

```
(TIMER  =  $5F5E100)
(TIMER  =  10000 * 10000)
(P100   =  10000 * 10000)
(TIMER  =  P100)
```

When the CNC is working in metric system (mm) resolution is in tenths of a micron, and figures are programmed in the format #5.4 (positive or negative, with 5 integers and 4 decimals), and if the CNC is operating in inches, resolution is in 0.00001 inches, figures being programmed with the format #4.5 (positive or negative, with 4 integers and 5 decimals).

For the convenience of the programmer, this control always allows the format #5.5 (positive or negative, with 5 integers and 5 decimals), adjusting each number appropriately to the working units every time they are used.

11.1.3 SYMBOLS

The symbols used in high-level language are:

( ) " = + - * / ,

11.2 VARIABLES

The internal CNC variables which can be accessed by high-level language are grouped in tables and can be read-only or read-write variables.

There is a group of mnemonics for showing the different fields of the table of variables. In this way, if it is required to access an element from one of these tables, the required field will be indicated by means of the corresponding mnemonic (for example TOR) and then the required element (TOR3).

The variables available at the CNC can be classified in the following way:

- General purpose parameters or variables
- Variables associated with tools.
- Variables associated with zero offsets.
- Variables associated with machine parameters
- Variables associated with work zones
- Variables associated with feedrates
- Variables associated with position coordinates
- Variables associated with the spindle
- Variables associated with the PLC
- Variables associated with local parameters
- Other variables

Variables which access to real values of the CNC interrupt the preparation of blocks and the CNC waits for each command to be performed before restarting block preparation.

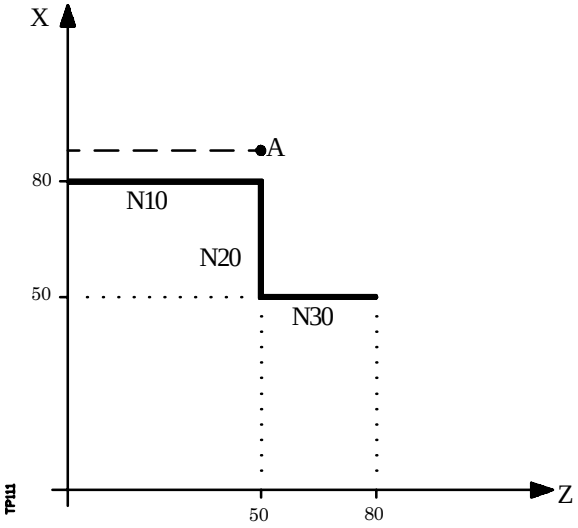
Thus, precaution must be taken when using this type of variable, as should they be placed between machining blocks which are working with compensation, undesired profiles may be obtained.

Example:

The following program blocks are performed in a section with G41 compensation.

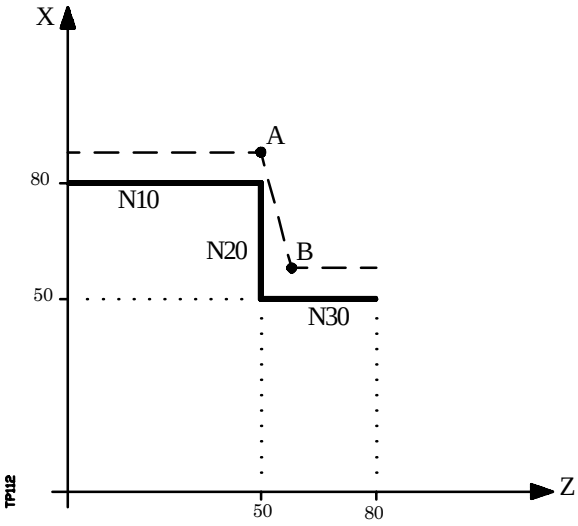
```
.......
.......
N10  X80  Z50
N15 (P100=POSX);     Assigns the value of the real coordinate in X to parameter P100
N20  X50  Z50
N30  X50  Z80
.......
.......
```

Block N15 interrupts block preparation and the execution of block N10 will finish at point A.



Once the execution of block N15 has ended, the CNC will continue block preparation from block N20 on.

As the next point corresponding to the compensated path is point "B", the CNC will move the tool to this point, executing path "A-B".



As can be observed, the resulting path is not the desired one, and therefore it is recommended to avoid the use of this type of variable in sections requiring tool compensation.

11.2.1 GENERAL PURPOSE PARAMETERS OR VARIABLES

The CNC has two types of general purpose variables: local parameters P0-P25 and global parameters P100-P299.

Programmers may use general purpose variables when editing their own programs. Later and during execution, the CNC will replace these variables with the values assigned to it at that time.

Example:

```
GP0 XP1 Y100                          —>  G1 X-12.5 Y100
(IF(P100*P101 EQ P102)GOTO N100)      —>  (IF(2*5 EQ 12)GOTO N100)
```

The use of these global purpose variables will depend on the type of block in which they are programmed and the channel of execution.

In block programmed in ISO code parameters can be associated with all fields, G X..C F S T D M. The block label number will be defined with a numerical value.

If parameters are used in blocks programmed in high-level language, these can be programmed within any expression.

Program which are executed in the user channel may contain any global parameter, but may not use local parameters.

The CNC will update the parameter table after processing the operations indicated in the block which is in preparation. This operation is always done before executing the block and for this reason, the values shown in the table do not necessarily have to correspond to the block being executed.

If the Execution Mode is abandoned after interrupting the execution of the program, the CNC will update the parameter tables with values corresponding to the block which was being executed.

When accessing the local parameter and global parameter table, the value assigned to each parameter may be expressed in decimal notation (4127.423) or in scientific notation (=23476 E-3).

The CNC has high level statements which allow the definition and use of subroutines which can be called from the main program, or from another subroutine, it also being possible to call a second subroutine, from the second to a third, etc. The CNC limits these calls, allowing up to a maximum of 15 nesting levels.

26 local parameters (P0-P25) can be assigned to a subroutine. These parameters which will be unknown for blocks external to the subroutine may be referenced by the blocks of this subroutine.

The CNC allows local parameters to be assigned to more than one subroutine, 6 nesting levels of local parameters being possible, within the 15 nesting levels of a subroutine.

Local parameters used in high-level language may be defined using the above format or by using the letter A-Z, except for N, so that A is equal to P0 and Z to P25.

The following example shows these two methods of definition:

```
(IF((P0+P1)   *  P2/P3   EQ  P4)   GOTO N100)
(IF((A+B)     *  C/D      EQ  E)    GOTO N100)
```

When using a parameter name (letter) for assigning a value to it (A instead of P0, for example), if the arithmetic expression is a constant, the statement can be abbreviated as follows:

(P0 = 13.7) —> (A = 13.7) —> (A13.7)

Be careful when using parenthesis since M30 is not the same as (M30). The CNC interprets (M30) as a high level statement meaning (P12 = 30) and not the execution of the miscellaneous M30 function.

The global parameter (P100-P299) can be used throughout the program by any block, irrespective of the nesting level.

Machining canned cycles (G81 thru G89) use the next local parameter nesting level when active.

11.2.2 VARIABLES ASSOCIATED WITH TOOLS

These variables are associated with the tool offset table, tool table and tool magazine table, so the values which are assigned to or read from these fields will comply with the formats established for these tables.

Tool offset table:

X, Z, R, I, K	Given in the active units: If G70, in inches. Max. ± 3937.00787 If G71, in millimeters. Max. ±99999.9999 If rotary, in degrees. Max. ±99999.9999
F	Integer value between 0 and 9.

Tool table

Tool offset number	0...NT OFFSET (maximum 255)
Family code	If normal tool, $0 \leq n < 200$ If special tool, $200 \leq n \leq 255$
Nominal life	0...65535 minutes or operations.
Real life	0.99999.99 minutes or 99999 operations
Cutter angle	In degrees. Max. 359.9999
Cutter width	Given in the active units: If G70, in inches. Max. ± 3937.00787 If G71, in millimeters. Max. ±99999.9999 If rotary, in degrees. Max. ±99999.9999
Cutting angle	In degrees. Max. 359.9999

Tool magazine table

Contents of each magazine position

	Tool number 1 ...NTOOL (maximum 255)
0	Empty
-1	Cancelled

Position of tool in magazine

	Position number 1 ..NPOCKET (maximum 255)
0	On spindle
-1	Not found
-2	In change position

Read-only variables

TOOL: Returns the active tool number
(P00 = TOOL); assigns the number of the active tool to P100

TOD: Returns the active tool offset number

NXTOOL: Returns the next tool number, selected but is awaiting the execution of M06 to be active.

NXTOD: Returns the number of the tool offset corresponding to the next tool, selected but is awaiting the execution of M06 to be active.

TMZPn: Returns the position occupied in the tool magazine by the indicated tool (n).

Read-write variables

TOXn: This variable allows reading or modifying the tool length value along the X axis assigned to the selected tool (n) in the offset table.

(P100 = TOX3); Assigns the X value of tool offset 3 to Parameter P100).
(TOR3 = P101); Assigns the value of parameter P101 to X of tool offset 3).

TOZn: This variable allows reading or modifying the tool length value along the Z axis assigned to the selected tool (n) in the offset table.

TOFn: This variable allows reading or modifying the value assigned to the location code of the selected tool (n) in the offset table.

TORn: This variable allows reading or modifying the value assigned to the Radius of the indicated tool offset (n) in the tool offset table.

TOIn: This variable allows reading or modifying the value assigned to the tool length wear along the X axis (I) of the indicated tool (n) in the tool offset table.

TOKn: This variable allows reading or modifying the value assigned to the tool length wear along the Z axis (K) of the indicated tool (n) in the tool offset table.

TLFDn: This variable allows the tool offset number of the indicated tool (n) to be read or modified on the tool table.

TLFFn: This variable allows the family code of the indicated tool (n) to be read or modified on the tool table.

TLFNn: This variable allows the value assigned as the nominal life of the indicated tool (n) to be read or modified on the tool table.

TLFRn: This variable allows the value corresponding to the real life of the indicated tool (n) to be read or modified on the tool table.

TMZTn: This variable allows the contents of the indicated position (n) to be read or modified on the tool magazine table.

NOSEAn: This variable allows reading or modifying the cutter angle value assigned to the indicated tool (n) in the tool offset table.

NOSEWn: This variable allows reading or modifying the cutter width value assigned to the indicated tool (n) in the tool offset table.

CUTAn: This variable allows reading or modifying the cutting angle value assigned to the indicated tool (n) in the tool offset table.

11.2.3 VARIABLES ASSOCIATED WITH ZERO OFFSETS

These variables are associated with the tool offset table, tool table and tool magazine table, so the values which are assigned to or read from these fields will comply with the formats established for these tables.

The zero offsets which are possible in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59.

The values for each axis are given in the active units:

If G70, in inches. Max. ± 3937.00787
If G71, in millimeters. Max. ±99999.9999
If rotary, in degrees. Max. ±99999.9999

Although there are variables which refer to each axis, the CNC only allows those referring to the selected axes in the CNC. Thus, if the CNC controls axes X, Y, Z, U and B, it only allows the variables ORGX, ORGY, ORGZ,. ORGU and ORGB in the case of ORG(X-C).

Read-only variables

ORG(X-C): Returns the value of the active zero offset in the selected axis. The value of the additive offset indicated by the PLC is not included in this value.

(P100 = ORGX); assigns to P100 the X value of the part zero active for the X axis. This value could have been set either by means of function G92 or by the variable "**ORG(X-C)n**".

PORGF: Returns the abscissa value of the polar origin with respect to the cartesian origin.

This coordinate value will be given in radius or diameter according to the setting of axis machine parameter "**DFORMAT**".

PORGS: Returns the ordinate value of the polar origin with respect to the cartesian origin.

This coordinate value will be given in radius or diameter according to the setting of axis machine parameter "**DFORMAT**".

Read-write variables

ORG(X-C)n: This variable allows the value of the selected axis to be read or modified on the table corresponding to the indicated zero offset (n).

(P110=ORGX55); Assigns the value of X to parameter P110 on the table corresponding to zero offset G55.

(ORGY 54=P111); Assigns the value of parameter P111 to the Y axis on the table corresponding to G54 zero offset.

PLCOF(X-C): This variable allows the value of the selected axis to be read or modified on the additive zero offset table indicated by the PLC.

If any of the PLCOF(X-C) variables are accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to begin block preparation again.

11.2.4 VARIABLES ASSOCIATED WITH MACHINE PARAMETERS

Variables associated with machine parameters are read-only variables.

In order to become familiar with the values returned it is advisable to consult the installation and start-up manual.

Values 1/0 correspond to the parameters which are defined with **YES/NO, +/-** and **ON/ OFF**.

The coordinates and feedrate values are given in the active units:
If G70, in inches. Max. ± 3937.00787
If G71, in millimeters. Max. ±99999.9999
If rotary, in degrees. Max. ±99999.9999

Read-only variables

MPGn: Returns the value assigned to the general machine parameter (n).

(P110=MPG 8);assigns the value of the general machine parameter "INCHES" to parameter P110, if millimeters P110=0 and if inches P110=1.

MP(X-C)n Returns the value which was assigned to the machine parameter (n) of the indicated axes.

(P110=MPY 1); assigns the value of the machine parameter P1 to arithmetic parameter P110 of the Y axis "DFORMAT", which indicates the format used in its display.

MPSn: Returns the value which was assigned to the main spindle machine parameter (n).

MPSSn: Returns the value which was assigned to the secondary spindle machine parameter (n).

MPASn: Returns the value of the machine parameter (n) for the auxiliary spindle.

MPLCn: Returns the value which was assigned to the PLC machine parameter (n)

11.2.5 VARIABLES ASSOCIATED WITH WORK ZONES

Variables associated with work zones are read-only variables.

The values for each axis are given in the active units:

If G70, in inches. Max. ± 3937.00787
If G71, in millimeters. Max. ±99999.9999
If rotary, in degrees. Max. ±99999.9999

The status of the work zones are defined according to the following code:

0 = Disabled.
1 = Enabled as no-entry zone.
2 = Enabled as no-exit zone.

Read-only variables

FZONE: Returns the status of work zone 1. (P100=FZONE); assigns to parameter P100 the status of work zone 1.

FZLO(X-C) Returns the value of the lower limit of Zone 1 according to the selected axis (X-C).

FZUP(X-C) Returns the value of the upper limit of Zone 1 according to the selected axis (X-C).

SZONE: Returns the status of work zone 2.

SZLO(X-C) Returns the value of the lower limit of Zone 2 according to the selected axis (X-C).

SZUP(X-C) Returns the value of the upper limit of Zone 2 according to the selected axis (X-C).

TZONE: Returns the status of work zone 3.

TZLO(X-C) Returns the value of the lower limit of Zone 3 according to the selected axis (X-C).

TZUP(X-C) Returns the value of the upper limit of Zone 3 according to the selected axis (X-C).

FOZONE: Returns the status of work zone 4.

FOZLO(X-C) Returns the value of the lower limit of Zone 4 according to the selected axis (X-C).

FOZUP(X-C) Returns the value of the upper limit of Zone 4 according to the selected axis (X-C).

11.2.6 VARIABLES ASSOCIATED WITH FEEDRATES

Read-only variables associated with real feedrate

FREAL: Returns the real feedrate of the CNC in mm/min. or inches/min.

(P100 = FREAL); Assigns the real feedrate value of the CNC to parameter P100

Read-only variables associated with function G94

FEED: Returns the feedrate selected in the CNC by means of the G94 function. This will be in mm/minute or inches/minute.

This feedrate can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCF: Returns the feedrate, in mm/minute or inches/minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCF: Returns the feedrate, in mm/minute or inches/minute, selected by PLC. If this has a value of 0 it means that it is not selected.

PRGF: Returns the feedrate, in mm/minute or inches/minute, selected by program.

Read-only variables associated with function G95

FPREV: Returns the feedrate selected in the CNC by means of the G95 function. This will be in mm/rev. or inches/rev.

This advance can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by DNC. If this has a value of 0 it means that it is not selected.

PLCFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by PLC. If this has a value of 0 it means that it is not selected.

PRGFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by program.

Read-only variables associated with function G32

PRGFIN: Returns the feedrate, in 1/min selected by program.

Also, the CNC variable FEED associated with G94 will show the resulting feedrate in mm/min or inches/min.

Read-only variables associated with Feedrate Override

FRO: Returns the Feedrate Override (%) selected at the CNC. This will be given by an integer between 0 and "MAXFOVR" (maximum 255).

This feedrate percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the switch.

DNCFRO: Returns the Feedrate Override % selected by DNC. If this has a value of 0 it means that it is not selected.

PLCFRO: Returns the Feedrate Override % selected by PLC. If this has a value of 0 it means that it is not selected.

CNCFR0: Returns the Feedrate Override % selected from the switch at the CNC Operator Panel.

PLCCFR: Returns the Feedrate Override % selected for the PLC execution channel.

Read-write variables

PRGFRO: This variable allows the feedrate percentage selected by program to be read or modified. This will be given by an integer between 0 and "MAXFOVR" (maximum 255). If it has a value of 0 this means that it is not selected.

(P110 = PRGFRO); assigns to P110 the % of feedrate override selected by program

(PFRGFRO = P111); sets the feedrate override % selected by program to the value of P111.

11.2.7 VARIABLES ASSOCIATED WITH COORDINATES

The values for each axis are given in the active units:
If G70, in inches. Max. ± 3937.00787
If G71, in millimeters. Max. ±99999.9999
If rotary, in degrees. Max. ±99999.9999

Read-only variables

PPOS(X-C): Returns the programmed theoretical coordinate of the selected axis.

(P100) = PPOSX); assigns to P100 the programmed theoretical position of the X axis.

POS(X-C) Returns the real coordinate of the selected axis referred to machine reference zero (home).

TPOS(X-C) Returns the theoretical coordinate (real + following error) of the selected axis referred to machine reference zero (home).

FLWE(X-C) Returns the amount of following error of the selected axis.

The coordinate values returned by PPOS(X-C), POS(X-C) and TPOS(X-C) will be expressed in the active units (radius or diameter) set by axis machine parameter "DFORMAT".

When accessing one of these variables (POS(X-C), TPOS(X-C) and FLWE(X-C)), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

DIST(X-C): These variables allow the distance travelled by the selected axis to be read or modified. This value is accumulative, is expressed in format 5.5 and is very useful when it is required to perform an operation which depends on the distance travelled by the axes, for example: in their lubrication.

(P100= DISTX) : assigns to P100 the distance travelled by the X axis

(DISTZ = P111); presets the variable indicating the distance travelled by the Z axis with the value of arithmetic parameter P111.

If any of the DIST(X-C) variables are accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

LIMPL(X-C):
LIMMI(X-C): With these variables, it is possible to set a second travel limit for each axis, LIMPL for the upper limit and LIMMI for the lower limit.

Since the second limits are activated or deactivated from the PLC, through general logic input ACTLIM2 (M5052), besides setting the limits, an auxiliary M code must be executed to let it know.

It is also recommended to execute function G4 after the change so the CNC executes the following blocks with the new limits.

The second travel limit will be taken into consideration when the first one has been set using axis machine parameters LIMIT+ (P5) and LIMIT- (P6).

11.2.8 VARIABLES ASSOCIATED WITH THE ELECTRONIC HANDWHEELS

Read-only variables

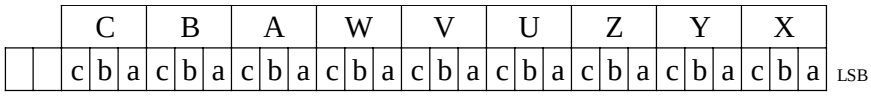
HANPF
HANPS
HANPT
HANPFO

They return the number of pulses of the first (HANPF), second (HANPS), third (HANPT) or fourth (HANPFO) handwheel received since the CNC was turned on. Regardless of whether the handwheel is connected to the AXES module or to the I/O module.

HANFCT

Returns the multiplying factor set from the PLC for each handwheel.

It must be used when having several electronic handwheels or when having a single handwheel but applying different multiplying factors (x1, x10, x100) to each axis.



Once the switch is position at one of the handwheel positions, the CNC checks this variable and depending on the values assigned to the "c b a" bits of each axis it applies the multiplying factor selected for each of them.

c b a	
0 0 0	The one indicated by the switch on the operator panel or keyboard
0 0 1	x1 Factor
0 1 0	x10 Factor
1 0 0	x100 Factor

If there are more than one bit to "1" on an axis, the least significant bit is taken into account.

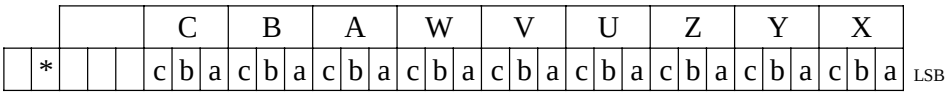
Thus: c b a	
1 1 1	x1 Factor
1 1 0	x10 Factor

Note: The screen always shows the value selected at the switch.

HBEVAR

It must be used when having a Fagor HBE handwheel.

It indicates whether the HBE handwheel is enabled or not, the axis to be jogged and the multiplying factor being applied (x1, x10, x100).



The CNC takes into account the reading of the HBE handwheel in JOG mode and if bit 30 (*) is enabled.

The values assigned to bits "c b a" indicate the axis to be jogged and the selected multiplying factor.

c	b	a	
0	0	0	It is not to be moved
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If there are several axes selected, the CNC attends to the next one in priority: X, Y, Z, U, V, W, A, B, C.

If there are more than one bit to "1" on an axis, the least significant bit is taken into account. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor

The HBE handwheel has priority. In other words, regardless of the mode selected at the CNC switch (continuous or incremental JOG, handwheel) HBEVAR is defined as other than "0", the CNC goes into handwheel mode.

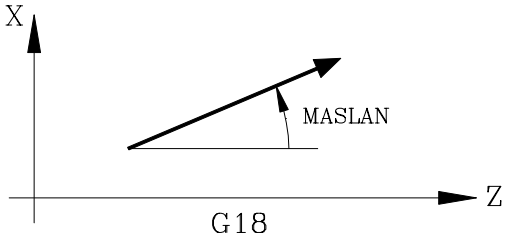
It highlights the selected axis and the multiplying factor selected by PLC. When the HBEVAR variable is set to "0", it displays the mode selected at the switch again.

For further information, refer to chapter 4 "Example of PLC program for Fagor HBE handwheel" in this manual.

Read and write variables

MASLAN must be used when the "Path Handwheel" mode is selected.

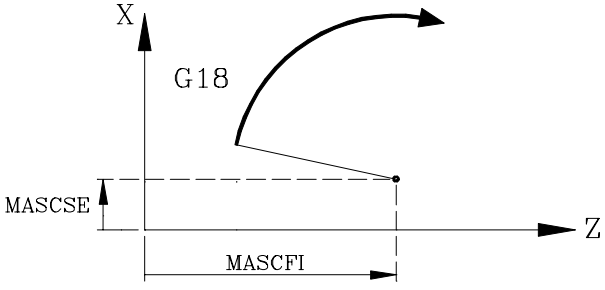
Indicates the angle of the linear path.



MASCFI
MASCSE

Must be used when the "Path Handwheel" mode is selected.

On circular paths, they indicate the coordinates of the arc center.



11.2.9 VARIABLES ASSOCIATED WITH THE MAIN SPINDLE

In these variables associated with the main spindle, their values are given in revolutions/ min and Override values for the main spindle are given in integers from 0 to 255.

Read-only variables

SREAL: Returns the real spindle turning speed in revolutions per minute.

(P100 = SREAL); assigns to P100 the real turning speed of the spindle.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

SPEED: Returns the spindle speed in revolutions per minute selected at the CNC.

This turning speed can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCS: Returns the turning speed in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCS: Returns the turning speed in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

PRGS: Returns the turning speed in revolutions per minute, selected by program.

CSS: Returns the Constant Surface Speed selected at the CNC. Its value is given in the active units (m/min. or feet/min.).

This C.S.S. may be indicated by the PLC or via DNC. The CNC will select one of them being the DNC the one with highest priority and the program the one with the lowest priority.

DNCCSS: Returns the Constant Surface Speed selected via DNC. Its value is given in m/min. or feet/min. A value of 0 means that no speed is selected.

PLCCSS: Returns the Constant Surface Speed selected via PLC. Its value is given in m/min. or feet/min. A value of 0 means that no speed is selected.

PRGCSS: Returns the Constant Surface Speed selected by program. Its value is given in m/min. or feet/min.

SSO: Returns the Override (%) of the spindle speed selected at the CNC. This will be given by an integer between 0 and "MAXSOVR" (maximum 255).

This spindle speed percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel.

DNCSSO: Returns the spindle speed percentage selected by DNC. If this has a value of 0 it means that it is not selected.

PLCSSO: Returns the spindle speed percentage selected by PLC. If this has a value of 0 it means that it is not selected.

CNCSSO: Returns spindle speed percentage selected from the front panel.

SLIMIT: Returns the value established for the spindle speed limit, in revolutions per minute, selected at the CNC.

This limit can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCSL: Returns the spindle speed limit in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCSL: Returns the spindle speed limit in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

PRGSL: Returns the spindle speed limit in revolutions per minute, selected by program.

POSS: Returns in degrees (max. ±99999.9999º) the real spindle position coordinate.

When accessing this variable, the CNC **interrupts block preparation** and it waits for this command to be executed before resuming block preparation.

RPOSS: Returns in degrees (between 0º and 360º) the real spindle position coordinate.

When accessing this variable, the CNC **interrupts block preparation** and it waits for this command to be executed before resuming block preparation.

TPOSS: Returns in degrees (max. ±99999.9999º) the theoretical spindle position coordinate (real + following error).

When accessing this variable, the CNC **interrupts block preparation** and it waits for this command to be executed before resuming block preparation.

RTPOSS: Returns in degrees (between 0º and 360º) the theoretical spindle position coordinate (real + following error).

When accessing this variable, the CNC **interrupts block preparation** and it waits for this command to be executed before resuming block preparation.

FLWES Returns in degrees (max. ±99999.9999º) the amount of following error of the spindle when it is in closed loop (M19).

When accessing this variable, the CNC **interrupts block preparation** and it waits for this command to be executed before resuming block preparation.

SYNCER Returns, in degrees (max. ±99999.9999), the following error of the second spindle with respect to the main spindle when they are synchronized in position.

Read-write variables

PRGSSO: This variable allows the percentage of the spindle speed selected by program to be read or modified. This will be given by an integer between 0 and "MAXSOVR" (maximum 255). If this has a value of 0 it means that it is not selected.

(P110 = PRGSSO); assigns to P110 the % of spindle speed selected by program.

(PRGSSO = P111); sets the value indicating spindle speed % selected by program to the value of arithmetic parameter P111.

11.2.10 VARIABLES ASSOCIATED WITH THE SECOND SPINDLE

In these variables associated with the second spindle, their values are given in revolutions/min and Override values for the second spindle are given in integers from 0 to 255.

Read-only variables

SSREAL: Returns the real spindle turning speed in revolutions per minute.

(P100 = SSREAL); assigns to P100 the real turning speed of the spindle.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

SSPEED: Returns the spindle speed in revolutions per minute selected at the CNC.

This turning speed can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

SDNCS: Returns the turning speed in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCS: Returns the turning speed in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

SPRGS: Returns the turning speed in revolutions per minute, selected by program.

SCSS: Returns the Constant Surface Speed selected at the CNC. Its value is given in the active units (m/min. or feet/min.).

This C.S.S. may be indicated by the PLC or via DNC. The CNC will select one of them being the DNC the one with highest priority and the program the one with the lowest priority.

SDNCCS: Returns the Constant Surface Speed selected via DNC. Its value is given in m/min. or feet/min. A value of 0 means that no speed is selected.

SPLCCS: Returns the Constant Surface Speed selected via PLC. Its value is given in m/min. or feet/min. A value of 0 means that no speed is selected.

SPRGCS: Returns the Constant Surface Speed selected by program. Its value is given in m/min. or feet/min.

SSSO: Returns the Override (%) of the spindle speed selected at the CNC. This will be given by an integer between 0 and "MAXSOVR" (maximum 255).

This spindle speed percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel.

SDNCSO: Returns the spindle speed percentage selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCSO: Returns the spindle speed percentage selected by PLC. If this has a value of 0 it means that it is not selected.

SCNCSO: Returns spindle speed percentage selected from the front panel.

SSLIMI: Returns the value established for the spindle speed limit, in revolutions per minute, selected at the CNC.

This limit can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

SDNCSL: Returns the spindle speed limit in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCSL: Returns the spindle speed limit in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

SPRGSL: Returns the spindle speed limit in revolutions per minute, selected by program.

SPOSS: Returns in degrees (max. ±99999.9999º) the real spindle position coordinate.

SRPOSS: Returns in degrees (between 0º and 360º) the real spindle position coordinate.

STPOSS: Returns in degrees (max. ±99999.9999º) the theoretical spindle position coordinate (real + following error).

SRTPOS: Returns in degrees (between 0º and 360º) the theoretical spindle position coordinate (real + following error).

SFLWES Returns in degrees (max. ±99999.9999º) the amount of following error of the spindle.

When accessing one of these variables (SPOSS, SRPOSS, STPOSS, SRTPOS or SFLWES), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

SPRGSO: This variable allows the percentage of the spindle speed selected by program to be read or modified. This will be given by an integer between 0 and "MAXSOVR" (maximum 255). If this has a value of 0 it means that it is not selected.

(P110 = SPRGSO); assigns to P110 the % of spindle speed selected by program.

(SPRGSO = P111); sets the value indicating spindle speed % selected by program to the value of arithmetic parameter P111.

11.2.11 VARIABLES ASOCIADAS A HERRAMIENTA MOTORIZADA

Read-only variables

ASPROG Must be used within the subroutine associated with function M45.
It returns the rpm programmed in M45 S
When programming just M45, the variable takes the value of "0".

This variable is updated just before executing function M45 so it is updated when executing the associated subroutine.

LIVRPM Must be used when operating in TC mode.
It returns the rpm selected by the operator for the live tool in TC mode.

11.2.12 VARIABLES ASSOCIATED WITH THE PLC

It should be borne in mind that the PLC has:

Inputs ..... (I1 thru I256)
Outputs ..... (O1 thru O256)
Marks ..... (M1 thru M5957)
Registers ... (R1 thru R499) of 32 bits each.
Timers ..... (T1 thru T256) with a timer count in 32 bits.
Counters ... (C1 thru C256) with a counter count in 32 bits.

If any variable is accessed which allows the status of a PLC variable to be read or modified (I,O,M,R,T,C), **block preparation is interrupted** and the CNC waits for this command to be executed in order to restart block preparation.

Read-only variables

PLCMSG: Returns the number of the active PLC message with the highest priority and will coincide with the number displayed on screen (1...128). If there is none, it returns 0.

(P100 = PLCMSG); assigns to P100 the number of the active PLC message with the highest priority.

Read-write variables

PLCIn: This variable allows 32 PLC inputs to be read or modified starting with the one indicated (n).

The value of the inputs which are used by the electrical cabinet cannot be modified as their values are determined by it. Nevertheless, the status of the remaining inputs can be modified.

PLCOn: This variable allows 32 PLC outputs to be read or modified starting from the one indicated (n).

(P110=PLC022); Assigns to P110 the value of PLC outputs 022 thru 053 (32 outputs).

(PLC22=SF); Sets outputs 022 thru 025 to 1 and outputs 026 thru 053 to 0.

Bit	31	30	29	28	27	26	25	24	23	22	21	20		6	5	4	3	2	1	0
	0	0	0	0	0	0	0	0	0	0	0	0		0	0	0	1	1	1	1
Output	53	52	51	50	49	48	47	46	45	44	43	42		28	27	26	25	24	23	22

PLCMn: This variable allows 32 PLC marks to be read or modified starting from the one indicated (n).

PLCRn: This variable allows the status of 32 register bits to be read or modified starting from the one indicated (n).

PLCTn: This variable allows the timer count to be read or modified starting from the one indicated (n).

PLCCn: This variable allows the counter count to be read or modified starting from the one indicated (n).

11.2.13 VARIABLES ASSOCIATED WITH LOCAL PARAMETERS

The CNC allows 26 local parameters (P0-P25) to be assigned to a subroutine, by using mnemonics PCALL and MCALL.

In addition to performing the required subroutine these mnemonics allow local parameters to be initialized.

Read-only variables

CALLP: Allows us to know which local parameters have been defined and which have not, in the call to the subroutine by means of the PCALL or MCALL mnemonic.

The information will be given in the 26 least significant bits (bits 0..25), each of these corresponding to the local parameter of the same number, as well as bit 12 corresponding to P12.

Each bit will indicate if the corresponding local parameter has been defined (=1) or not (=0).

31	30	29	28	27	26	25	24	23	22	21	20		6	5	4	3	2	1	0
0	*0*	*0*	*0*	*0*	*0*	*	*	*	*	*	*		*	*	*	*	*	*	*

Example:

```
(PCALL 20, P0=20, P2=3, P3=5)      ;  Call to subroutine 20.
....
....
(SUB 20)                           ;  Beginning of subroutine 20
(P100=CALLP)
....
....
```

In parameter P100 the following will be obtained:

0000	0000	0000	0000	0000	0000	0000	1101

LSB

11.2.14 SERCOS VARIABLES

They are used for data exchange between the CNC and the servo drives via Sercos interface.

Read-only variables

TSVAR(X-C) identifier ..... for the axes
TSVARS identifier ..... for the main spindle
TSSVAR identifier ..... for the second spindle

It returns the third attribute of the sercos variable corresponding to the "identifier". The third attribute is used in particular software applications and its information is coded according to the Sercos standard.

(P110=SVARX 40) assigns to parameter P110 the third attribute of the sercos variable of identifier 40 of the X axis which corresponds to "VelocityFeedback"

Write-only variables

SETGE(X-C) ..... for the axes
SETGES ..... for the main spindle
SSETGS ..... for the second spindle

The drive may have up to 8 work ranges or gears (0 through 7). Sercos identifier 218, GearRatioPreselection.

It may also have up to 8 parameter sets (0 through 7). Sercos indentifier 217, ParameterSetPreselection.

These variables permit changing the work range (gear) or the paramete set for each drive.

The 4 least significant bits of these variables must indicate the work gear and the 4 most significant bits the parameter set to be selected.

Read-Write variables

SVAR(X-C) identifier ..... for the axes
SVARS identifier ..... for the main spindle
SSVAR identifier ..... for the second spindle

They permit reading or modifying the value of the sercos variable corresponding to the axis identifier.

(P110=SVARX 40) assigns to parameter P110 the value of the sercos variable of identifier 40 of the X axis which corresponds to the "VelocityFeedback"

11.2.15 OTHER VARIABLES

Read-only variables

OPMODE: Returns the code corresponding to the selected operating Mode.

0 = Main menu.

10 = Automatic execution.
11 = Single block execution.
12 = MDI in EXECUTION
13 = Tool inspection

20 = Theoretical path movement simulation
21 = G functions simulation
22 = G, M, S and T functions simulation
23 = Simulation with movement on main plane
24 = Simulation with rapid movement

30 = Normal editing
31 = User editing
32 = TEACH-IN editing
33 = Interactive editor
34 = Profile editor

40 = Movement in continuous JOG
41 = Movement in incremental JOG
42 = Movement with electronic handwheel
43 = HOME search in JOG
44 = Position preset in JOG
45 = Tool calibration
46 = MDI in JOG
47 = JOG user operation

50 = Zero offset table
51 = Tool Offset table
52 = Tool table
53 = Tool magazine table
54 = Global parameter table
55 = Local parameter table

60 = Utilities

70 = CNC status
71 = DNC status

80 = Editing PLC files
81 = Compiling PLC program
82 = PLC monitoring
83 = Active PLC messages
84 = Active PLC pages (screens)
85 = Save PLC program
86 = Restore PLC program
87 = “PLC resources in use” mode
88 = PLC statistics

90 = Graphic Editor

100 = General machine parameter table
101 = Axis machine parameter tables
102 = Spindle machine parameter tables
103 = Serial port machine parameter tables
104 = PLC machine parameter table
105 = M function table
106 = Spindle and cross compensation table
107 = Table of machine parameters for Ethernet

110 = Diagnosis: configuration
111 = Diagnosis: hardware test
112 = Diagnosis: RAM memory test
113 = Diagnosis: flash memory test
114 = User diagnosis
115 = Hard Disk diagnosis

OPMODA Indicates the operating mode currently selected when working with the main channel.

Use the OPMODE variable to know at any time the selected operating mode (main channel, user channel, PLC channel).

This information is given at the least significant bits with a "1" when active and with a "0" when not active or when it is not available in the current version.

bit 0 Program in execution.
bit 1 Program in simulation.
bit 2 Block in execution via MDI, JOG
bit 3 Repositioning in progress.
bit 4 Program interrupted, by CYCLE STOP
bit 5 MDI, JOG Block interrupted
bit 6 Repositioning interrupted
bit 7 In tool inspection
bit 8 Block in execution via CNCEX1
bit 9 Block via CNCEX1 interrupted
bit 10 CNC ready to accept JOG movements: jog, handwheel, teach-in, inspection.
bit 11 CNC ready to receive the CYCLE START command: execution, simulation and MDI modes.
bit 12 The CNC is not ready to execute anything involving axis or spindle movement.

OPMODB Indicates the type of simulation currently selected. This information is given at the least significant bits with a "1" indicating the currently selected one.

bit 0 Theoretical path
bit 1 G functions
bit 2 G M S T functions
bit 3
bit 4 Rapid
bit 5 Rapid (S=0)

OPMODC Indicates the axes selected by Handwheel. This information is given at the least significant bits indicating with a "1" the one currently selected.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The axis number corresponds to the order it is programmed.
Example: If the CNC controls the X, Y, Z, U, B, C axes, Axis 1 will be the X axis, Axis 2= Y, Axis 3=Z, Axis 4= U, Axis 5= B, Axis 6= C.

NBTOOL Indicates the tool number being managed.

Example: There is a manual tool changer. Tool T1 is currently selected and the operator requests tool T5.

The subroutine associated with the tools may contain the following instructions:

(P103 = NBTOOL)
(MSG "SELECT T?P103 AND PRESS CYCLE START")

Instruction (P103 = NBTOOL) assigns the number of the tool currently being managed to parameter P103. Therefore, P103=5

The message displayed by the CNC will be ""SELECT T5 AND PRESS CYCLE START".

PRGN: Returns the program number being executed. Should none be selected, a value of -1 is returned.

BLKN: Returns the label number of the last block executed.

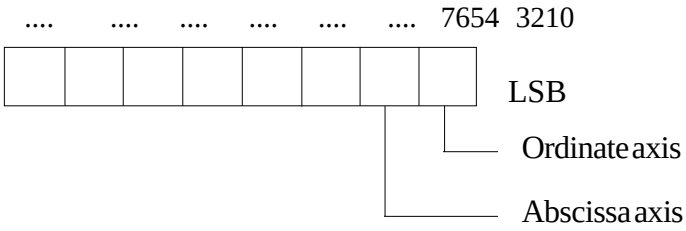
GSn: Returns the status of the G function indicated (n). 1 if it is active and 0 if not.

(P120=GS17); assigns the value 1 to parameter P120 if the G17 function is active and 0 if not.

MSn: Returns the status of the M function indicated (n). 1 if it is active and 0 if not.

This variable provides the status of M00, M01, M02, M03, M04, M05, M06, M08, M09, M19, M30, M41, M42, M43, M44 and M45 functions.

PLANE: Returns data on the abscissa axis (bits 4 to 7) and the ordinate axis (bits 0 to 3) of the active plane in 32 bits and in binary.



The axes are coded in 4 bits and indicate the axis number according to the programming order.

Example: If the CNC controls the X,Y,Z,U,B,C axes and is selected in the ZX plane (G18).

(P122 = PLANE) assigns value $31 to parameter P122.

0000	***0000***	***0000***	***0000***	***0000***	***0000***	***0011***	***0001***

LSB

MIRROR Returns in the least significant bits in a group of 32 bits, the status of the mirror image of each axis, 1 in the case of being active and 0 if not.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The name of the axis corresponds to the order they have been programmed.

Example: If the CNC controls axes X, Y, Z, U, B, C Axis 1=X, Axis2=Y, Axis3=Z, Axis4=U, Axis5=B, Axis6=C.

SCALE: Returns the general scaling factor being applied.

SCALE(X-C): Returns the indicated specific scaling factor of the axis (X-C).

PRBST: Returns the status of the probe.

0 = The probe is not touching the part.
1 = The probe is touching the part.

CLOCK: Returns in seconds the time indicated by the system clock. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

TIME: Returns the time in hours-minutes-seconds format.

(P150=TIME); assigns hh-mm-ss to P150. For example if the time is 18h 22m 34 sec., P150 will contain 182234.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

DATE: Returns the date in year-month-day format.

(P151=DATE); assigns year-month-day to P151. For example if the date is April 25th 1992, P151 will contain 920425.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

CYTIME: Returns in hundredths of a second the time it has taken to make the part. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

FIRST: Indicates whether it is the first time that a program has been run. It returns a value of 1 if it is the first time and 0 for the remainder of times.

A first-time execution is considered as being one made:

After turning on the CNC.
After pressing the "Shift-Reset" keys.
Every time a new program is selected.

ANAIn: Returns in volts and in #1.4 format (values #5 Volts), the status of the analog input indicated (n), it being possible to select one among eight (1...8) analog inputs.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

AXICOM Returns in the 3 least significant bytes the axis pairs toggled with function G28.

		Pair 3		Pair 2		Pair 1		
		Axis 2	Axis 1	Axis 2	Axis 1	Axis 2	Axis 1	LSB

The axes are coded in 4 bits and indicate the axis number (1 through 7) according to the order they are programmed.

If the CNC controls the X, Z, C, W axes and G28 ZW has been programmed, the AXICOM variable will show the following information:

		Pair 3		Pair 2		Pair 1		
						W	Z	
0000	0000	0000	0000	0000	0000	0100	0010	LSB

TANGAN Variable associated with tangential control, G45. It indicates the programmed angular position.

Read-write variables

TIMER: This variable allows time, in seconds, indicated by the clock enabled by the PLC to be read or modified. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

PARTC: The CNC has a part counter whose count increases every time M30 or M02 is executed and this variable allows it value to be read or modified, which will be given by a number between 0 and 4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

KEY: Returns the code of the last key accepted.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

KEYSRC: This variable allows the origin of keys to be read or modified, possible values being:

0 = Keyboard
1 = PLC
2 = DNC

The CNC only allows modification of this variable if this is at 0.

ANAOn: This variable allows the required analog output (n) to be modified. The value assigned will be expressed in volts and in the ±2.4 format (±10 Volts).

The analog outputs which are free among the eight (1..8) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in an occupied one.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

11.3 CONSTANTS

Constants are defined as being all those fixed values which cannot be altered by a program. The following are considered as constants:

- Numbers expressed in the decimal system.
- Hexadecimal numbers.
- PI (¶) constant.
- Read-only tables and variables as their value cannot be altered with a program.

11.4 OPERATORS

An operator is a symbol which indicates mathematical or logic manipulations which must bemade. The CNC has arithmetic, relational, logic, binary, trigonometric operators and special operators.

Arithmetic operators

+	: add.	P1 = 3 + 4	—> P1 = 7
-	: subtraction, also to indicate a negative number	P2 = 5 - 2 P3 = -(2 3)	—> P2 = 3 —> P3 = -6
*	: multiplication	P4 = 2 3*	--> P4 = 6
/	: division	P5 = 9*/ 2	—> P5 = 4.5
MOD	: module (remainder of a division)	P6 = 7 **MOD** 4	—> P6 = 3
EXP	: exponential	P7 = 2 **EXP** 3	—> P7 = 8

Relational operators

EQ	: equal
NE	: different
GT	: greater than
GE	: greater than or equal to
LT	: less than
LE	: less than or equal to

Logic or binary operators

NOT, OR, AND, XOR: act as logic operators between conditions and as binary operators between variables and constants.

```
IF (FIRST AND GS1 EQ 1) GOTO N100
P5 = (P1 AND (NOT P2 OR P3))
```

Trigonometric functions

SIN	: sine	P1 = **SIN** 30	—>	P1 = 0.5
COS	: cosine	P2 = **COS** 30	—>	P2 = 0.8660
TAN	: tangent	P3 = **TAN** 30	—>	P3 = 0.5773
ASIN	: arc sine	P4 = **ASIN** 1	—>	P4 = 90
ACOS	: arc cosine	P5 = **ACOS** 1	—>	P5 = 0
ATAN	: arc tangent	P6 = **ATAN** 1	—>	P6 = 45
ARG	: ARG (x,y) arc tangent y/x	P7 = **ARG**(-1,-2)	—>	P7 = 243.4349

There are two functions for calculating the arc tangent: ATAN which returns the result between ±90° and ARG given between 0 and 360°.

Other functions

ABS	: absolute value	P1 = **ABS** -8	—>	P1 = 8
LOG	: decimal logarithm	P2 = **LOG** 100	—>	P2 = 2
SQRT	: square root	P3 = **SQRT** 16	—>	P3 = 4
ROUND	: rounding up a number	P4 = **ROUND** 5.83	—>	P4 = 6
FIX	: integer	P5 = **FIX** 5.423	—>	P5 = 5
FUP	: if integer takes integer	P6 = **FUP** 7	—>	P6 = 7
	if not, takes entire part + 1	P6 = **FUP** 5.423	—>	P6 = 6

BCD : converts given number to BCD P7 = **BCD** 234 —> P7 = 564

0010	0011	0100

BIN : converts given number to binary P8 = **BIN** $AB —> P8 = 171

1010	1011

Conversions to binary and BCD are made in 32 bits, it being possible to represent the number 156 in the following formats:

Decimal	156
Hexadecimal	9C
Binary	0000 0000 0000 0000 0000 0000 1001 1100
BCD	0000 0000 0000 0000 0000 0001 0101 0110

11.5 EXPRESSIONS

An expression is any valid combination between operators, constants and variables.

All expressions must be placed between brackets, but if the expression is reduced to an integer, the brackets can be removed.

11.5.1 ARITHMETIC EXPRESSIONS

These are formed by combining functions and arithmetic, binary and trigonometric operators with the constants and variables of the language.

The way to operate with these expressions is established by operator priorities and their associativity:

Priority from highest to lowest	**Associativity**
NOT, functions, - (negative)	from right to left
EXP, MOD	from left to right
*,/	from left to right
+,-(add, subtract)	from left to right
relational operators	from left to right
AND, XOR	from left to right
OR	from left to right

It is advisable to use brackets to clarify the order in which the evaluation of the expression is done.

(P3 = P4/P5 - P6 * P7 - P8/P9)
(P3 = (P4/P5)-(P6*P7)-(P8/P9))

The use of repetitive or additional brackets will not produce errors nor will they slow down execution.

In functions, brackets must be used except when these are applied to a numerical constant, in which case they are optional.

(SIN 45) (SIN (45))	both are valid and equivalent.
(SIN 10+5)	the same as ((SIN 10)+5).

Expressions can be used also to reference parameters and tables:

(P100 = P9)
(P100 = P(P7))
(P100 = P(P8 + SIN (P8 * 20)))
(P100 = ORGX 55)
(P100 = ORGX (12+P9))
(PLCM5008 = PLCM5008 OR 1); selects Single Block execution (M5008=1)
(PLCM5010 = PLCM5010 AND $FFFFFFFE); Frees feedrate Override (M5010=0)

11.5.2 RELATIONAL EXPRESSIONS

These are arithmetic expressions joined by relational operators

(IF (**P8 EQ 12.8**) ... ; Analyzes if the value of P8 is equal to 12.8
(IF (**ABS(SIN(P24)) GT SPEED**) ... ;Analyzes if the sine is greater than the spindle speed.
(IF (**CLOCK LT(P9*10.99**)) ... ; Analyzes if the clock count is less than (P9*10.99)

At the same time these conditions can be joined by means of logic operators.

(IF((**P8EQ12.8) OR(ABS(SIN(P24))GT SPEED)) AND (CLOCK LT(PT*10.99**)) ...

The result of these expressions is true or false.

12. *PROGRAM CONTROL STATEMENTS*

The control statements available to high-level programming can be grouped as follows:

*** Programming statements** consisting of:

Assignment statements
Display statements
Enable-disable statements
Flow control statements
Subroutine statements
Statements for generating programs
Screen customizing statements

*** Screen customizing statements**

Only one statement can be programmed in each block, and no other additional information may be programmed in this block.

12.1 ASSIGNMENT STATEMENTS

This is the simplest type of statement and can be defined as:

(target=arithmetic expression)

A local or global parameter or a read-write variable may be selected as target. The arithmetic expression may be as complex as required or a simple numerical constant.

(P102 = FZLOX)
(ORGX 55 = (ORGX 54 + P100))

In the specific case of designating a local parameter using its name (A instead of P0, for example) and the arithmetic expression being a numerical constant, the statement can be abbreviated as follows:

(P0=13.7) ==> (A=13.7) ==> (A13.7)

Within a single block, up to 26 assignments can be made to different targets, a single assignment being interpreted as the set of assignments made to the same target.

(P1=P1+P2,P1=P1+P3,P1=P*P4,P1=P1/p5) is the same as (P1=(P1+P2+P3)*P4/P5).

The different assignments which are made in the same block will be separated by commas ",".

12.2 DISPLAY STATEMENTS

(ERROR integer, “error text”)

This statement stops the execution of the program and displays the indicated error, it being possible to select this error in the following ways:

(ERROR integer). This will display the error number indicated and the text associated to this number according to the CNC error code (should there be one).

(ERROR integer “error text”). This will display the number and the error text indicated, it being necessary to write the text between quote marks “”.

(ERROR “error text”). This will display the error text only.

The error number may be defined by means of a numerical constant or an arithmetic parameter. When using a local parameter, its numeric format must be used (P0 thru P25 instead of A thru Z).

Programming Examples:

(ERROR 5)
(ERROR P100)
(ERROR “Operator error”)
(ERROR 3, “Operator error”)
(ERROR P120, “Operator error)

(MSG “message”)

This statement will display the message indicated between quote marks.

The CNC screen is provided with an area for displaying DNC or user program messages, and always displays the last message received irrespective of where it has come from.

Example:

(MSG “Check tool”)

(DGWZ expression 1, expression 2, expression 3, expression 4)

The DGWZ instruction (Define Graphic Work Zone) defines the graphics area.

Each expression forming the instruction syntax correspond to one of the limits and they must be defined in millimeters or inches.

expression 1 X minimum
expression 2 X maximum
expression 3 Y minimum
expression 4 Y maximum

12.3 ENABLING-DISABLING STATEMENTS

(ESBLK and DSBLK)

After executing the mnemonic **ESBLK**, the CNC executes all the blocks which come after as if it were dealing with a single block.

This single block treatment is kept active until it is cancelled by executing the mnemonic **DSBLK**.

In this way, should the program be executed in the SINGLE BLOCK operating mode, the group of blocks which are found between the mnemonics ESBLK and DSBLK will be executed in a continuous cycle, i.e., execution will not be stopped at the end of a block but will continue by executing the following one.

Example:

```
G01 X30 Z10  F1000  T1   D1
(ESBLK)                        ; Start of single block
G01 X20 Z10
G01 X20 Z20
G02 X10 Z30  I-10   K0
(DSBLK)                        ; Cancellation of single block
G01 X10 Z40
M30
```

(ESTOP and DSTOP)

After executing the mnemonic **DSTOP**, the CNC enables the Stop key, as well as the Stop signal from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic **ESTOP**.

(EFHOLD and DFHOLD)

After executing the mnemonic **DFHOLD**, the CNC enables the Feed-Hold input from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic **EFHOLD**.

12.4 FLOW CONTROL STATEMENTS

The **GOTO** and **RPT** statements cannot be used in programs executed from a PC connected to one of the serial lines.

(GOTO N(expression)

The mnemonic **GOTO** causes a jump within the same program, to the block defined by the label N(expression).

The execution of the program will continue after the jump, from the indicated block.

The jump label can be addressed by means of a number or by any expression which results in a number.

Example:

```
      G00 X30 Z10 T2      D4
          X30 Z20
      (GOTO N22)                    ; Jump statement
          X20 Z20                   ; Is not executed
          X20 Z10                   ; Is not executed
N22   G01 X10 Z10 F10000            ; Continues execution in this block
      G02 X0  Z40 I-10    K0
      .....
      .....
```

(RPT N(expression), N(expression))

The mnemonic **RPT** executes, within the same program, the part of the program which exists between the blocks defined by means of the labels **N(expression)**.

Both labels can be indicated by means of a number or by any expression which results in a number.

The part of the program selected by means of the two labels must belong to the same program, by first defining the initial block and then the final block.

The execution of the program will continue in the block following the one in which the mnemonic RPT was programmed, once the selected part of the program has been executed.

Example:

```
N10 G91 G01 X-20  Z-20
        X10
    G02 X10 Z-10  I10   K0
N20 G01     Z-10
N30 (RPT N10, N20)  N3
N40 G01 X20
    M30
```

When reaching block N30, the program will execute section N10-N20 three times. Once this has been completed, the program will continue execution in block N40.

(IF condition <action1> ELSE <action2>)

This statement analyzes the given condition which must be a relational expression. If the condition is true (result equal to 1), **<action1>** will be executed, otherwise (result equal to 0) **<action2>** will be executed.

Example:

(IF(P8 EQ 12.8) CALL 3 ELSE PCALL 5, A2, B5, D8)

If P8 = 12.8 executes the mnemonic (CALL3)
If P8 <> 12.8 executes the mnemonic (PCALL 5, A2, B5, D8)

The statement can lack the ELSE part, i.e., it will be enough to program IF condition <action1>.

Example:

(IF(P8 EQ 12.8)CALL 3)

Both <action1> and <action2> can be expressions or statements, except for mnemonics IF and SUB.

Due to the fact that in a high level block local parameters can be named by means of letters, expressions of this type can be obtained:

(IF (E EQ 10)M10)

If the condition of parameter P5 (E) having a value of 10 is met, the miscellaneous function M10 will not be executed, since a high level block cannot have ISO code commands. In this case M10 represents the assignment of value 10 to parameter P12, i.e., one can program either:

(IF(E EQ 10)M10) or (IF(P5 EQ 10) P12=10)

12.5 SUBROUTINE STATEMENTS

A subroutine is a part of a program which, being properly identified, can be called from any position of a program to be executed.

A subroutine can be kept in the memory of the CNC as an independent part of a program and be called one or several times, from different positions of a program or different programs.

Only subroutines stored in CNC RAM memory can be executed. To execute a subroutine stored in the memory card, the hard disk or in a PC connected through the serial line, it must be copied first into the RAM memory of the CNC.

If the subroutine is too large to be copied into RAM memory, turn the subroutine into a program and use the EXEC statement as described in section 12.6

(SUB integer)

The mnemonic **SUB** defines the set of program blocks which are programmed after this block as a subroutine by identifying this subroutine with an integer, between 0 and 9999, which is specified after it:

There can not be two subroutines with the same identification number in the CNC memory, even when they belong to different programs.

(RET)

The mnemonic **RET** indicates that the subroutine which was defined by the mnemonic SUB, finishes in this block.

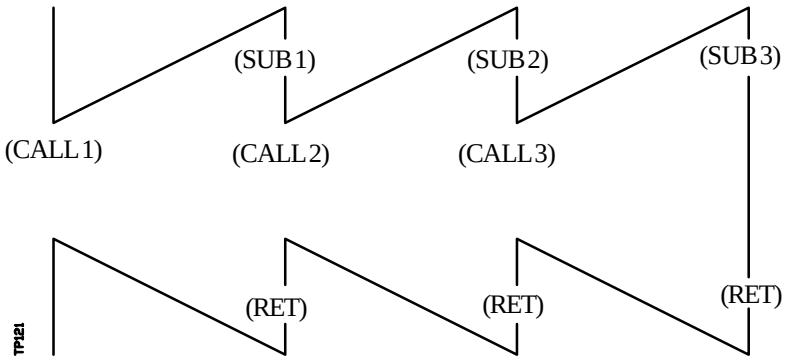
Example:

```
(SUB 12)                ;  Definition of subroutine 12
G91 G01 XP0   F5000
        ZP1
        X-P0
        Z-P1
(RET)                   ;  End of subroutine
```

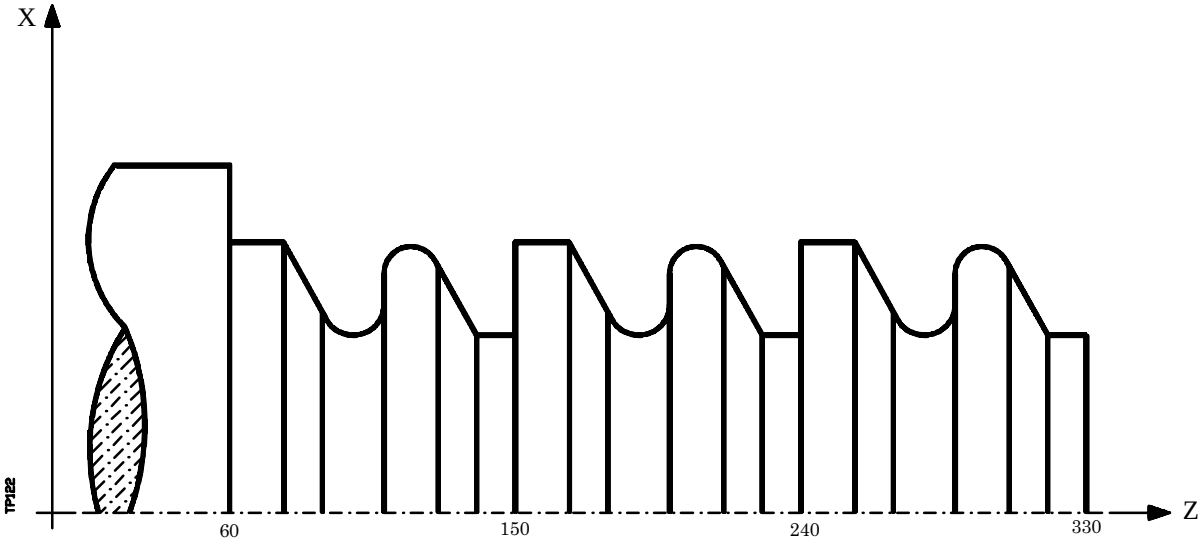
(CALL (expression))

The mnemonic **CALL** makes a call to the subroutine indicated by means of a number or by means of any expression which results in a number.

As a subroutine may be called from a main program, or a subroutine, from this subroutine to a second one, from the second to a third, etc..., the CNC limits these calls to a maximum of 15 nesting levels, it being possible to repeat each of the levels 9999 times.



Example



G90 G01 X100 Z330
(CALL 10)
G90 G01 X100 Z240
(CALL 10)
G90 G01 X100 Z150
(CALL 10)
M30

(SUB 10)
G91 G01 Z-10
X40 Z-10
G03 X0 Z-20 I0 K-10
G01 X-20
G02 X0 Z-20 I0 K-10
G01 X40 Z-10
Z-20
(RET)

(PCALL (expression), (assignment statement), (assignment statement),...)

The mnemonic **PCALL** calls the subroutine indicated by means of a number or any expression which results in a number. In addition, it allows up to a maximum of 26 local parameters of this subroutine to be initialized.

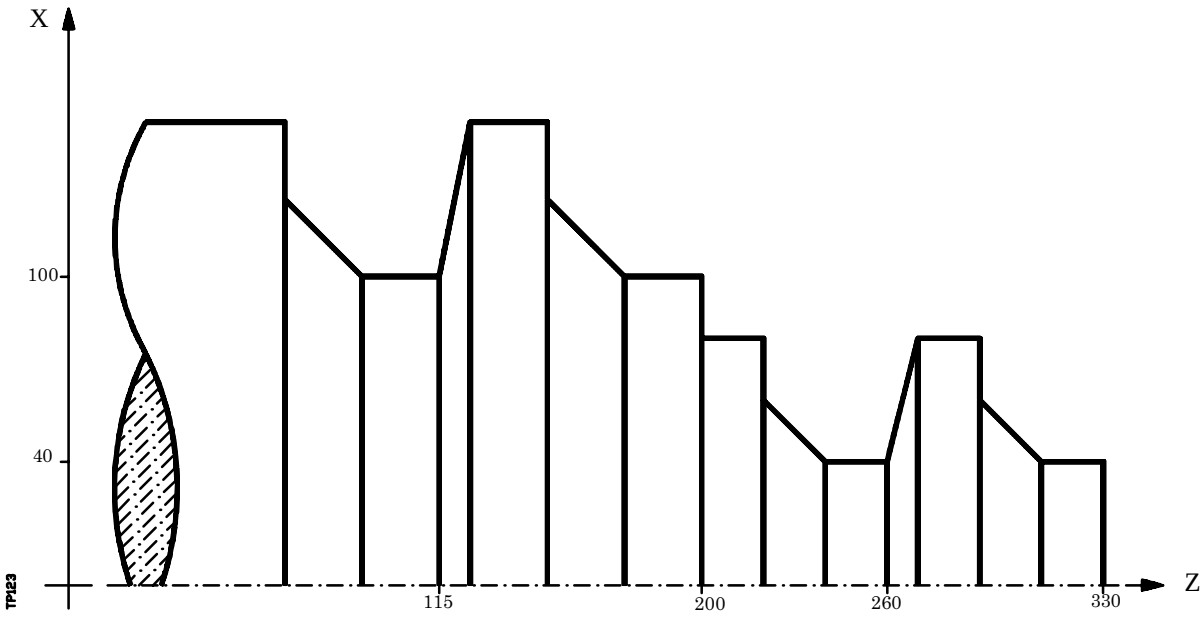
These local parameters are initialized by means of **assignment statements.**

Example: (PCALL 52, A3, B5, C4, P10=20)

In this case, in addition to generating a new subroutine nesting level, a new local parameter nesting level will be generated, there being a maximum of 6 levels of local parameter nesting, within the 15 levels of subroutine nesting.

Both the main program and each subroutine which is found on a parameter nesting level, will have 26 local parameters (P0-P25).

Example:



```
G90  G01  X80    Z330
(PCALL 10, P0=20, P1=-10)     ;  or also (PCALL 10, A20, B10)
G90  G01  X80    Z260
(PCALL 10, P0=20 P1=-10)      ;  or also (PCALL 10, A20 B-10)
G90  G01  X200   Z200
(PCALL 10, P0=30 P1=-15)      ;  or also (PCALL 10, A30 B-15)
G90  G01  X200   Z115
(PCALL 10, P0=30 P1=-15)      ;  or also (PCALL 10, A30 B-15)
M30

(SUB 10)
G91  G01  ZP1
     XP0  ZP1
     XP0
     ZP1
(RET)
```

(MCALL (expression), (assignment statement), (assignment statement),...)

By means of the mnemonic **MCALL**, any user-defined subroutine (SUB integer) acquires the category of canned cycle.

The execution of this mnemonic is the same as the mnemonic PCALL, but the call is modal, i.e., if another block with axis movement is programmed at the end of this block, after this movement, the subroutine indicated will be executed and with the same call parameters.

If, when a modal subroutine is selected, a movement block with a number of repetitions is executed, for example X10 N3, the CNC will execute the movement only once (X10) and after the modal subroutine, as many times as the number of repetitions indicates.

Should block repetitions be chosen, the first execution of the modal subroutine will be made with updated call parameters, but not for the remaining times, which will be executed with the values which these parameters have at that time.

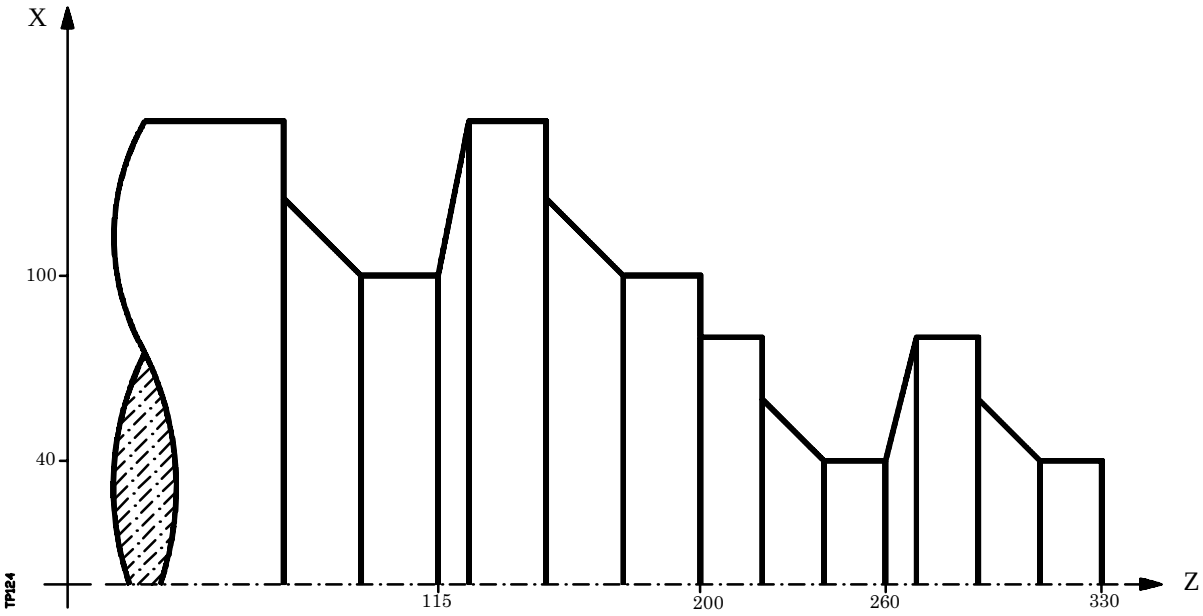
If, when a subroutine is selected as modal, a block containing the MCALL mnemonic is executed, the present subroutine will lose its modal quality and the new subroutine selected will be changed to modal.

(MDOFF)

The mnemonic **MDOFF** indicates that the modal quality acquired by the subroutine with the MCALL mnemonic, finishes in this block.

The use of modal subroutines simplifies programming.

Example:




```
(P100 = 20, P101 =-10)
G90  G01  X80    Z330
(MCALL 10)
G90  G01  X80    Z260
(P100 = 30, P101 = -15)
G90  G01  X200   Z200
G90  G01  X200   Z115
(MDOFF)
M30

(SUB 10)
G91  G01      ZP101
       XP100  ZP101
       XP100
       ZP101
(RET)
```

(PROBE (expression), (assignment statement), (assignment statement),...)

The mnemonic **PROBE** calls the probe cycle indicated by means of a number or any **expression** which results in a number. In addition, it allows the local parameters of this subroutine to be initialized by means of **assignment statements.**

This mnemonic also generates a new level of subroutine nesting.

12.5.1 INTERRUPTION SUBROUTINE STATEMENTS

Whenever one of the general interruption logic input is activated, "INT1" (M5024), "INT2" (M5025), "INT3" (M5026) or "INT4 (M5027), the CNC temporarily interrupts the execution of the program in progress and starts executing the interruption subroutine whose number is indicated by the corresponding general parameter.

With INT1 (M5024) the one indicated by machine parameter INT1SUB (P35)
With INT2 (M5025) the one indicated by machine parameter INT2SUB (P36)
With INT3 (M5026) the one indicated by machine parameter INT3SUB (P37)
With INT4 (M5027) the one indicated by machine parameter INT4SUB (P38)

The interruption subroutines are defined like any other subroutine by using the statements: "(SUB integer)" and "(RET)".

The interruption subroutines do not change the level of the local arithmetic parameters; thus they can only contain global arithmetic parameters.

Within an interruption subroutine, it is possible to use the "(REPOS X, Y, Z, ...)" statement described next.

Once the execution of the subroutine is over, the CNC resumes the execution of the program which was interrupted.

(REPOS X, Y, Z, ...)

The REPOS statement must always be used inside an interruption subroutine and facilitates the repositioning of the machine axes to the point of interruption.

When executing this statement, the CNC moves the axes to the point where the program was interrupted.

* The axes are repositioned one at a time.

* It is not necessary to define all the axes, only those to be repositioned.

* The axes forming the main plane move together; thus, it is not required to program both axes since the CNC moves both of them with the first one. The movement is not repeated when defining the second one, it is ignored.

Example: The main plane is formed by the X and Y axes, the Z axis is the longitudinal (perpendicular) axis and the machine uses the C and W axes as auxiliary axes. It is desired to first move the C axis, then the X and Y axes and finally the Z axis..

This repositioning move may be defined in any of the following ways:

(REPOS C, X, Y, Z) **(REPOS C, X, Z)** **(REPOS C, Y, Z)**

If the REPOS statement is detected while executing a subroutine not activated by an interruption input, the CNC will issue the corresponding error message.

12.6 PROGRAM STATEMENTS

With this CNC, from a program in execution one can:

Execute another program ..... Statement (EXEC P.....)
Generate a new program ..... Statement (OPEN P.....)
Add blocks to an existing program ..... Statement (WRITE.....)

(EXEC P(expression), (directory)

The **EXEC P** statement executes the part-program of the indicated directory

The part-program may be defined by a number or any **expression** resulting in a number.

By default, the CNC assumes that the part-program is in the CNC's RAM memory. If it is in another device, it must be indicated in (**directory**).

CARDA	in the "Memkey CARD"
HD	on the hard disk
DNC1	at a PC connected through serial line 1
DNC2	at a PC connected through serial line 2

(OPEN P(expression), (destination directory), A/D, "program comment")

This statement starts editing a part-program whose number will be given by any number or **expression** resulting in a number.

By default, the new part-program edited will be stored in the CNC's RAM memory. To store it another device, it must be indicated in (**destination directory**).

CARDA	in the "Memkey CARD"
HD	on the hard disk
DNC1	at a PC connected through serial line 1
DNC2	at a PC connected through serial line 2

Parameter **A/D** is used when the program to be edited already exists.

A	The CNC appends the new blocks after the ones already existing
D	The CNC deletes the existing program and starts editing a new one.

A **program comment** may also be associated with it. This comment will later be displayed next to it on the program directory.

The OPEN statement is very useful when digitizing parts because it allows generating a program from a program already in execution. That generated program will depend on the values assumed by the program being executed.

To edit blocks, the WRITE statement must be used as described next.

Notes: If the program to be edited already exists and the A/D parameters are not defined, the CNC will display an error message when executing the block.

The program opened with the OPEN statement is closed when executing an M30, or another OPEN statement and after an Emergency or Reset.

From a PC, only programs stored in the CNC'S RAM memory, in the CARD A, or in the Hard Disk module can be opened

(WRITE <block text>)

The mnemonic **WRITE** adds, after the last block of the program which began to be edited by means of the mnemonic OPEN P, the information contained in **<block text>** as a new program block.

If parametric programming is used within the <block text> and it has been edited in ISO code, when executing the block, all the parameters (global and local) are replaced with the value they have at time. Blocks edited in high level language are NOT replaced.

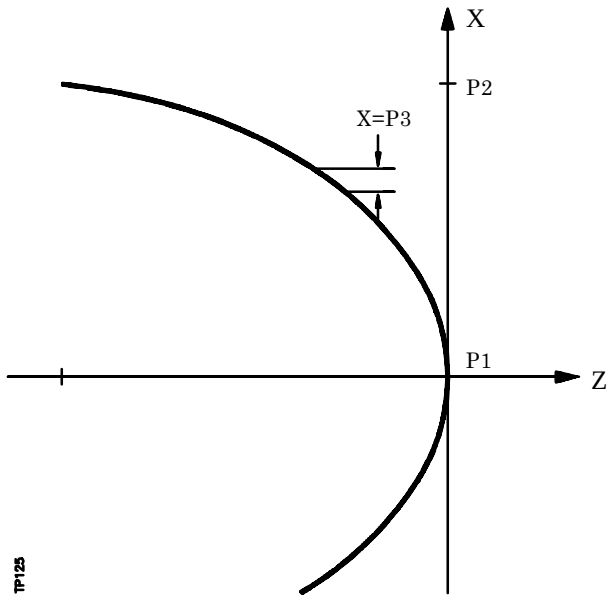
In the customizing programs edited by the operator, the IB instruction may be used so the blocks edited in high level assume the parameter value.

Examples for P100=10, P101=20 and P102=55

```
(WRITE G1 XP100 YP101 F100)          =>  G1 X10 Y20 F100
(WRITE (IF (P100 EQ P101) CALL 3))   =>  (IF (P100 EQ P101) CALL 3)
(WRITE (SUB P102))                   =>  (SUB P102)
(IB1=(P102))
(WRITE (SUB (IB1))                   =>  (SUB 55)
```

If the mnemonic WRITE is programmed without having programmed the mnemonic OPEN previously, the CNC will display the corresponding error, except when editing a user customized program, in which case a new block is added to the program being edited.

Example of the creation of a program which contains several points of a parabolic path whose formula is: $Z = -K X^2$



The X axis is programmed in diameter and subroutine 2 is used whose parameters mean the following:

Call parameters:

A	or P0	Value of constant "K"
B	or P1	X coordinate of initial point
C	or P2	X coordinate of final point
D	or P3	incremental step along X

Calculated parameters:

E	or P4	X coordinate
F	or P5	Z coordinate

One way of using this example could be:

```
G00 X0 Z0
(PCALL 2, A0.01, B0, C100, D1)
M30
```

Program generating subroutine:

```
        (SUB 2)
        (OPEN P12345)          ; Starts the generation of program P12345
        (P4 = P1)              ; X = initial X
N100    (IF (P4+P3 GE P2) P4=P2 ELSE P4=P4+P3)
        (P5=-(P0*P4*P4))
        (WRITE G01 XP4 ZP5)    ; Movement block
        (IF (P4 NE P2) GOTO N100)
        (WRITE M30)            ; End of program block
        (RET)
```

12.7 SCREEN CUSTOMIZING STATEMENTS (GRAPHIC EDITOR)

Customizing statements may be used only when customizing programs made by the user.

These customizing programs must be stored in CNC RAM memory and may utilize the "Programming Statements" and they will be executed in the special channel designed for this use. The program selected in each case will be indicated in the following general machine parameters.

In “USERDPLY” the program required to execute in the Execution Mode will be indicated.

In “USEREDIT” the program required to execute in the Editing Mode will be indicated.

In “USERMAN” the program required to execute in the Manual (JOG) Mode will be indicated.

In “USERDIAG” the program required to execute in the Diagnosis Mode will be indicated.

The customizing programs may have up to five nesting levels besides their current one. Also, the customizing statements do not allow local parameters, nevertheless all global parameters may be used to define them.

(PAGE (expression))

The mnemonic **PAGE** displays the page number indicated by means of a number or by means of any **expression** which results in a number.

User-defined pages will be from page 0 to page 255 and will be defined from the CNC keyboard in the Grahic Editor mode and as indicated in the Operating Manual.

System pages will be defined by a number greater than 1000. See the corresponding appendix.

(SYMBOL (expression 1), (expression 2), (expression 3))

The mnemonic **SYMBOL** displays the symbol whose number is indicated by means of the value of **expression 1** once this has been evaluated.

Its position on screen is also defined by **expression 2** (column) and by **expression 3** (row).

Expression 1, expression 2 and expression 3 may contain a number or any expression which results in a number.

The CNC allows to display any user-defined symbol (0-255) defined at the CNC keyboard in the Graphic Editor mode such as is indicated in the Operating Manual.

In order to position it within the display area its pixels must be defined, 0-639 for columns (expression 2) and 0-335 for rows (expression 3).

(IB (expression) = INPUT “text”, format))

The CNC has 26 data entry variables (IBO-1B25)

The **IB** mnemonic displays the text indicated in the data input window and stores the data input by the user in the entry variable indicated by means of a number or by means of any **expression** which results in a number.

The wait for data entry will only occur when programming the format of the requested data. This **format** may have a sign, integer part and decimal part.

If it bears the “minus” sign, it will allow positive and negative values, and if it does not have a sign, it will only allow positive values.

The integer part indicates the maximum number of digits (0-6) desired to the left of the decimal point.

The decimal part indicates the maximum number of digits (0-5) desired to the right of the decimal point.

If the numerical format is not programmed; for example (IB1=INPUT "text"), the mnemonic will only display the indicated text without waiting for the data to be entered.

(ODW (expression 1), (expression 2), (expression 3))

The mnemonic **ODW** defines and draws a white window on the screen with fixed dimensions (1 row and 14 columns).

Each mnemonic has an associated number which is indicated by the value of **expression 1** once this has been evaluated.

Likewise, its position on screen is defined by **expression 2** (row) and by **expression 3** (column).

Expression 1, expression 2 and expression 3 may contain a number or any expression which results in a number.

The CNC allows 26 windows (0-25) to be defined and their positioning within the display area, providing 21 rows (0-20) and 80 columns (0-79).

(DW(expression 1) = (expression 2), DW (expression 3) = (expression 4),...)

The mnemonic **DW** displays in the window indicated by the value for expression 1, expression 3, .. once they have been evaluated, the numerical data indicated by expression 2, expression 4, ...

Expression 1, expression 2, expression 3, ... may contain a number or any expression which may result in a number.

The following example shows a dynamic variable display:

```
        (ODW 1,6,33)                    ;  Defines data window 1
        (ODW 2,14,33)                   ;  Defines data window 2
N10  (DW1=DATE,DW2=TIME)                ;  Displays the date in window 1 and
                                           the time in 2
        (GOTO N10)
```

The CNC allows displaying the data in decimal, hexadecimal and binary format. The following instructions are available:

(DW1 = 100) Decimal format. Value "100" displayed in window 1.
(DWH2 = 100) Hexadecimal format. Value "64" displayed in window 2.
(DWB3 = 100) Binary format. Value "01100100" displayed in window 3.

When using the binary format, the display is limited to 8 digits in such a way that a value of "11111111" will be displayed for values greater than 255 and the value of "10000000" for values more negative than -127.

Besides, the CNC allows the number stored in one of the 26 data input variables (IB0-IB25) to be displayed in the requested window.

The following example shows a request and later display of axis feedrate.

```
(ODW3,4,60)                     ; Defines data window 3
(IB1=INPUT"Axis feed:",5.4)     ; Axis feedrate request
(DW3=IB1)                       ; Displays feedrate in window 3
```

(SK(expression 1) = “text1” (expression 2) = “text 2”, ...)

The mnemonic **SK** defines and displays the new softkey menu indicated.

Each of the **expressions** will indicate the softkey number which it is required to modify (1-7, starting from the left) and the **texts** which it is required to write in them.

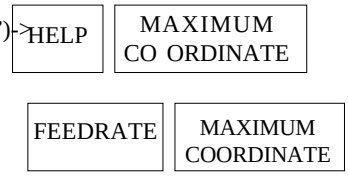
Expression 1, expression 2, expression 3, ... may contain a number or any expression which may result in a number.

Each text will allow a maximum of 20 characters which will be shown on two lines of 10 characters each. If the text selected has less than 10 characters, the CNC will center it on the top line, but if it has more than 10 characters the programmer will center it.

Examples:

(SK 1=”HELP”,SK 2=”MAXIMUM COORDINATE”) —>

(SK 1=”FEEDRATE”,SK 2=”_MAXIMUM__COORDINATE”)->



Warning:

If while a standard CNC softkey menu is active, one or more softkeys are selected via high level language statement: “**SK**”, the CNC will clear all existing softkeys and it will only show the selected ones.

If while a user softkey menu is active, one or more softkeys are selected via high level language statement “**SK**”, the CNC will only replace the selected softkeys leaving the others intact.

(WKEY)

The mnemonic **WKEY** stops execution of the program until the key is pressed.

The pressed key will be recorded in the KEY variable.

Example

```
....
....
(WKEY)                              ; Wait for key
(IF KEY EQ $FC00 GOTO N1000)        ; If key F1 has been pressed,
                                    continue in N1000
....
....
```

(WBUF “text”, (expression))

The **WBUF** statement can only be used when editing a program in the user channel.

This mnemonic may be programmed in two ways:

(WBUF “text”, (expression))

This statement adds the text and value of the expression once this has been evaluated, to the block which is being edited and within the data input window.

(Expression) may contain a number or any expression which results in a number.

It will be optional to program the expression, but it will be required to define the text. If no text is required, “” must be programmed.

(WBUF)

Enters into memory, adding to the program being edited and after the cursor position, the block being edited by means of (WBUF "text", (expression)). It also clears the editing buffer in order to edit a new block.

This allows the user to edit a complete program without having to quit the user editing mode after each block and press ENTER to "enter" it into memory.

Example:

(WBUF”(PCALL 25,”)	; Adds “(PCALL 25,” to the block being edited
(IB1=INPUT “Parameter A:”,-5.4)	; Request of Parameter A
(WBUF “A=”,IB1)	; Adds “A=(value entered) to the block being edited.
(IB2=INPUT”Parameter B:”,-5.4)	; Request of Parameter B
(WBUF”,B=”,IB2	; Adds “B=(value entered)” to the block being edited
(WBUF”)”)	; Adds “)” to the block being edited
(WBUF)	; Enters the edited block into memory

After executing this program the block being edited contains:

(PCALL 25, A=23.5, B=-2.25)

(SYSTEM)

The mnemonic **SYSTEM** stops execution of the user customized program and returns to the corresponding standard menu of the CNC.

Customizing program example

The following customizing program must be selected as user program associated to the Editing Mode.

After selecting the Editing Mode and pressing the USER softkey, this program starts executing and it allows assisted editing of 2 user cycles. This editing process is carried out a cycle at a time and as often as desired.

```
; Displays the initial editing page (screen)
N0   (PAGE 10)

; Sets the softkeys to access the various modes and requests a choice
     (SK 1="CICLO 1",SK 2="CICLO 2",SK 7="SALIR")
N10  (WKEY)                                        ;Request a key
     (IF KEY EQ $FC00 GOTO N10)                    ;Cycle 1
     (IF KEY EQ $FC01 GOTO N20)                    ;Cycle 2
     (IF KEY EQ $FC06 SYSTEM ELSE GOTO N10)        ;Quit or request a key

; CYCLE 1
; Displays page 11 and defines 2 data entry windows
N10  (PAGE 11)
     (ODW 1,10,60)
     (ODW 2,15,60)

;Editing
     (WBUF "(PCALL 1,")      ; Adds (PCALL 1,  to the block being edited

     (IB 1=INPUT "X:",-6.5)  ; Requests the value of X
     (DW 1=IB1)              ; Data window 1 shows the entered value
     (WBUF "X",IB1)          ; Adds X (entered value) to the block being edited

     (WBUF ",")              ; Adds ,  to the block being edited

     (IB 2=INPUT "Y:",-6.5)  ; Requests the value of Y
     (DW 2=IB2)              ; Data window 2 shows the entered value
     (WBUF "Y",IB2)          ; Adds Y (entered value) to the block being edited

     (WBUF ")")              ; Adds )  to the block being edited

     (WBUF)                  ; Enters the edited block into memory. For example: (PCALL 1, X2, Y3)

     (GOTO N0)
```

;(This sample program continues on next page)

*; **CYCLE 2***

; Displays page 12 and defines 3 data entry windows

```
N20  (PAGE 12)
     (ODW 1,10,60)
     (ODW 2,13,60)
     (ODW 3,16,60)

;Editing
     (WBUF "( PCALL 2,")      ; Adds (PCALL 2, to the block being edited

     (IB 1=INPUT "A:",-6.5)   ; Requests the value of A
     (DW 1=IB1)               ; Data window 1 shows the entered value
     (WBUF "A",IB1)           ; Adds A (entered value) to the block being edited

     (WBUF ",")               ; Adds ,  to the block being edited

     (IB 2=INPUT "B:",-6.5)   ; Requests the value of B
     (DW 2=IB2)               ; Data window 2 shows the entered value
     (WBUF "B",IB2)           ; Adds B (entered value) to the block being edited

     (WBUF ",")               ; Adds ,  to the block being edited

     (IB 3=INPUT "C:",-6.5)   ; Requests the value of C
     (DW 3=IB3)               ; Data window 3 shows the entered value
     (WBUF "C",IB3)           ; Adds C (entered value) to the block being edited

     (WBUF ")")               ; Adds )  to the block being edited

     (WBUF)                   ;Enters the edited block into memory. Example: (PCALL 2, A3, B1, C3)

     (GOTO N0)
```

APPENDIX

A. ISO CODE PROGRAMMING 3

B. INTERNAL CNC VARIABLES 5

C. HIGH LEVEL PROGRAMMING 11

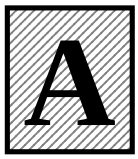
D. KEY CODES 13

E. LOGIC OUTPUTS FOR KEY CODE STATUS 14

F. KEYS INHIBITING CODES 15

G. PROGRAMMING ASSISTANCE SYSTEM PAGES 16

H. MAINTENANCE 19



ISO CODE PROGRAMMING

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid travel	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise circular interpolation	6.3
G03	*		*	Counter-clockwise circular interpolation	6.3
G04				Dwell/block preparation stop	7.1, 7.2
G05	*	?	*	Round corner	7.3.2
G06			*	Absolute arc center coordinates	6.4
G07	*	?		Square corner	7.3.1
G08			*	Arc tangent to previous path	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G15	*		*	C axis	6.13
G16	*		*	Selection of main plane in two directions	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.7.1
G21				Definition of upper work zone limits	3.7.1
G22			*	Activate/cancel work zones	3.7.2
G28	*		*	Selection of the second spindle or axis toggle	5.4, 7.8
G29	*	*		Selection of the main spindle or axis toggle	5.4, 7.8
G30	*		*	Spindles synchronized in position	5.5
G32	*		*	Feedrate "F" as an inverted function of time	6.15
G33	*		*	Threadcutting	6.12
G36			*	Automatic radius blend	6.10
G37			*	Tangential entry	6.8
G38			*	Tangential exit	6.9
G39			*	Automatic chamfer blend	6.11
G40	*	*		Cancellation of tool radius compensation	8.1
G41	*		*	Right-hand tool radius compensation	8.1
G42	*		*	Left-hand tool radius compensation	8.1
G45	*		*	Tangential control	6.16
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look ahead	7.4
G52			*	Movement to hard stop	6.14
G53			*	Program coordinates with respect to home	4.3
G54	*		*	Absolute zero offset	4.4.2
G55	*		*	Absolute zero offset	4.4.2
G56	*		*	Absolute zero offset	4.4.2
G57	*		*	Absolute zero offset	4.4.2
G58	*		*	Additive zero offset	4.4.2
G59	*		*	Additive zero offset	4.4.2

Function	M	D	V	Meaning	Section
G60			*	Axial drilling / tapping canned cycle	9.13
G61			*	Radial drilling / tapping canned cycle	9.14
G62			*	Longitudinal slot milling canned cycle	9.15
G63			*	Radial slot milling canned cycle	9.16
G66			*	Pattern repeat canned cycle	9.1
G68			*	Stock removal canned cycle along X axis	9.2
G69			*	Stock removal canned cycle along Z axis	9.3
G70	*	?	*	Programming in inches	3.3
G71	*	?		programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G74			*	Machine reference search	4.2
G75			*	Probing until touching	10.1
G76			*	Probing while touching	10.1
G77	*		*	Slaved axis	7.7.1
G77S	*		*	Spindle synchronism in speed	5.5
G78	*	*		Slaved axis cancellation	7.7.2
G78S	*	*		Spindle synchronism cancellation	5.5
G81			*	Turning canned cycle with straight sections	9.4
G82			*	Facing canned cycle with straight sections	9.5
G83			*	Drilling canned cycle	9.6
G84			*	Turning canned cycle with circular sections	9.7
G85			*	Facing canned cycle with circular sections	9.8
G86			*	Longitudinal threadcutting canned cycle	9.9
G87			*	Face threadcutting canned cycle	9.10
G88			*	Grooving canned cycle along X axis	9.11
G89			*	Grooving canned cycle along Z axis	9.12
G90	*	?		Programming in absolute	3.4
G91	*	?	*	Programming in incremental	3.4
G92				Coordinate preset/spindle speed limit	4.4.1
G93				Polar origin	4.5
G94	*	?		Feedrate in millimeters(inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters(inches) per revolution	5.2.2
G96	*		*	Constant Surface Speed	5.3.1
G97	*	*		Spindle speed in r.p.m.	5.3.2

M means MODAL, i.e., that once programmed, the G function remains active as long as another incompatible G function is not programmed, M02, M30, EMERGENCY, RESET are not programmed or the CNC is not turned on or off.

Letter **D** means BY DEFAULT, i.e., that these will be assumed by the CNC when turned on, after executing M02, M30 or after EMERGENCY or RESET.

In cases indicated with **?** it must be interpreted that the BY DEFAULT of these G functions depends on the settings of the general CNC machine parameters.

V means that the G function is displayed next to the machining conditions in the execution and simulation modes.

B INTERNAL CNC VARIABLES

R indicates that the variable can be read.
W indicates that the variable can be modified.

VARIABLES ASSOCIATED WITH TOOLS

(Section 11.2.2)

Variable	CNC	PLC	DNC	
TOOL	R	R	R	Number of active tool.
TOD	R	R	R	Number of active tool offset.
NXTOOL	R	R	R	Number of the next requested tool waiting for M06.
NXTOD	R	R	R	Number of the next tool's offset.
TMZPn	R	R	-	(n) tool's position in the tool magazine.
TLFDn	R/W	R/W	-	(n) tool's offset number.
TLFFn	R/W	R/W	-	(n) tool's family code.
TLFNn	R/W	R/W	-	Nominal life assigned to tool (n).
TLFRn	R/W	R/W	-	Real life value of tool (n).
TMZTn	R/W	R/W	-	Contents of tool magazine position (n).
TOXn	R/W	R/W	-	Tool length offset (n) along X axis.
TOZn	R/W	R/W	-	Tool length offset (n) along Z axis.
TOFn	R/W	R/W	-	Location code (F) of offset (n).
TORn	R/W	R/W	-	Tool radius (R) value of offset (n).
TOIn	R/W	R/W	-	Tool length wear (I) of offset (n) along X axis.
TOKn	R/W	R/W	-	Tool length wear (K) of offset (n) along Z axis.
NOSEAn	R/W	R/W	-	Cutter angle of the selected tool (n).
NOSEWn	R/W	R/W	-	Cutter width of the selected tool (n).
CUTAn	R/W	R/W	-	Cutting angle of the selected tool (n).

VARIABLES ASSOCIATED WITH ZERO OFFSETS

(Section 11.2.3)

Variable	CNC	PLC	DNC	
ORG(X-C)	R	R	-	Zero offset active on the selected axis without including the incremental offset activated via PLC.
PORGF	R	-	R	Abscissa coordinate value of polar origin.
PORGS	R	-	R	Ordinate coordinate value of polar origin.
ORG(X-C)n	R/W	R/W	R	Zero offset (n) value of the selected axis.
PLCOF(X-C)	R/W	R/W	R	Value of the additive Zero Offset activated via PLC.

VARIABLES ASSOCIATED WITH MACHINE PARAMETERS

(Section 11.2.4)

Variable	CNC	PLC	DNC	
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to machine parameter (n) of the axis (X-C)
MPSn	R	R	-	Value assigned to machine parameter (n) of the main spindle.
MPSSn	R	R	-	Value assigned to machine parameter (n) of the second spindle.
MPASn	R	R	-	Value assigned to machine parameter (n) of the auxiliary spindle.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

VARIABLES ASSOCIATED WITH THE WORK ZONES

(Section 11.2.5)

Variable	CNC	PLC	DNC	
FZONE	R	R/W	R	Status of work zone 1.
FZLO(X-C)	R	R/W	R	Lower limit of work zone 1 along the selected axis (X/C).
FZUP(X-C)	R	R/W	R	Upper limit of work zone 1 along the selected axis (X/C).
SZONE	R	R/W	R	Status of work zone 2.
SZLO(X-C)	R	R/W	R	Lower limit of work zone 2 along the selected axis (X/C).
SZUP(X-C)	R	R/W	R	Upper limit of work zone 2 along the selected axis (X/C)
TZONE	R	R/W	R	Status of work zone 3.
TZLO(X-C)	R	R/W	R	Lower limit of work zone 3 along the selected axis (X/C).
TZUP(X-C)	R	R/W	R	Upper limit of work zone 3 along the selected axis (X/C).
FOZONE	R	R/W	R	Status of work zone 4.
FOZLO(X-C)	R	R/W	R	Lower limit of work zone 4 along the selected axis (X/C).
FOZUP(X-C)	R	R/W	R	Upper limit of work zone 4 along the selected axis (X/C).

VARIABLES ASSOCIATED WITH FEEDRATES

(Section 11.2.6)

Variable	CNC	PLC	DNC	
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.

Variables associated with function G94

FEED	R	R	R	Active feedrate at the CNC in mm/min or inch/min.
DNCF	R	R	R/W	Feedrate selected via DNC.
PLCF	R	R/W	R	Feedrate selected via PLC.
PRGF	R	R	R	Feedrate selected by program.

Variables associated with function G95

FPREV	R	R	R	Active feedrate at CNC, in m/rev or inch/rev.
DNCFPR	R	R	R/W	Feedrate selected via DNC.
PLCFPR	R	R/W	R	Feedrate selected via PLC.
PRGFPR	R	R	R	Feedrate selected by program.

Variables associated with function G32

PRGFIN	R	R	R	Feedrate selected by program. In 1/min.

Variables associated with Feedrate Override

FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Feedrate Override (%) selected by program.
DNCFRO	R	R	R/W	Feedrate Override (%) selected by DNC.
PLCFRO	R	R/W	R	Feedrate Override (%) selected by PLC.
CNCFRO	R	R	R	Feedrate Override (%) selected from the front panel knob.
PLCCFR	R	R/W	R	Feedrate Override (%) of the PLC execution channel

VARIABLES ASSOCIATED WITH POSITION VALUES *(Section 11.2.7)*

Variable	CNC	PLC	DNC	
PPOS(X-C)	R	-	-	Theoretical programmed position value (coordinate).
POS(X-C)	R	R	R	Real position value of the indicated axis.
TPOS(X-C)	R	R	R	Theoretical (real + lag) position value of the indicated axis.
FLWE(X-C)	R	R	R	Following error of the indicated axis.
DIST(X-C)	R/W	R/W	R	Distance travelled by the indicated axis.
LIMPL(X-C)	R/W	R/W	R	Upper second travel limit.
LIMMI(X-C)	R/W	R/W	R	Lower second travel limit.

VARIABLES ASSOCIATED WITH HANDWHEELS ***(Section 11.2.8)***

Variable	CNC	PLC	DNC	
HANPF	R	-	-	1st handwheel pulses received since CNC power-up.
HANPS	R	-	-	2nd handwheel pulses received since CNC power-up.
HANPT	R	-	-	3rd handwheel pulses received since CNC power-up.
HANPFO	R	-	-	4th handwheel pulses received since CNC power-up.
HANFCT	R	R/W	R	Multiplying factor different per handwheel. When having several.
HBEVAR	R	R/W	R	HBE handwheel. Reading enabled, axis to jog, x factor (x1, x10, x100)
MASLAN	R/W	R/W	R/W	Angle of the linear path with "Path Handwheel".
MASCFI	R/W	R/W	R/W	Arc center coordinates when using "Path Handwheel".
MASCSE	R/W	R/W	R/W	Arc center coordinates when using "Path Handwheel".

VARIABLES ASSOCIATED WITH THE MAIN SPINDLE

(Section 11.2.9)

Variable	CNC	PLC	DNC	
SREAL	R	R	R	Real spindle speed in r.p.m.
SPEED	R	R	R	Active spindle speed at the CNC.
DNCS	R	R	R/W	Spindle speed selected via DNC.
PLCS	R	R/W	R	Spindle speed selected via PLC.
PRGS	R	R	R	Spindle speed selected by program.
CSS	R	R	R	Constant surface feed active at the CNC in meters/min or feet/min.
DNCCSS	R	R	R/W	Constant surface feed selected via DNC.
PLCCSS	R	R/W	R	Constant surface feed selected via PLC.
PRGCSS	R	R	R	Constant surface feed selected by program.
SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
PRGSSO	R/W	R	R	Spindle Speed Override (%) selected by program.
DNCSSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
PLCSSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
CNCSSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SLIMIT	R	R	R	Spindle speed limit, in rpm, active at the CNC.
DNCSL	R	R	R/W	Spindle speed limit selected via DNC.
PLCSL	R	R/W	R	Spindle speed limit selected via PLC.
PRGSL	R	R	R	Spindle speed limit selected by program.
POSS	R	R	R	Spindle real position, (max ±999999999 in 0.0001º)
RPOSS	R	R	R	Spindle real position, (between 0º and 360º in 0.0001º)
TPOSS	R	R	R	Spindle theoretical position, (max ±999999999 in 0.0001º)
RTPOSS	R	R	R	Spindle theoretical position (between 0º and 360º in 0.0001º)
FLWES	R	R	R	Spindle following error when in Closed loop (M19), in degrees
SYNCER	R	R	R	Following error of the second spindle with respect to the main spindle when they're both synchronized.

VARIABLES ASSOCIATED WITH THE SECOND SPINDLE

(Section 11.2.10)

Variable	CNC	PLC	DNC	
SSREAL	R	R	R	Real spindle speed in r.p.m.
SSPEED	R	R	R	Active spindle speed at the CNC.
SDNCS	R	R	R/W	Spindle speed selected via DNC.
SPLCS	R	R/W	R	Spindle speed selected via PLC.
SPRGS	R	R	R	Spindle speed selected by program.
SCSS	R	R	R	Constant surface feed active at the CNC in meters/min or feet/min.
SDNCCS	R	R	R/W	Constant surface feed selected via DNC.
SPLCCS	R	R/W	R	Constant surface feed selected via PLC.
SPRGCS	R	R	R	Constant surface feed selected by program.
SSSO	R	R	R	Spindle Speed Override (%) active at the CNC.
SPRGSO	R/W	R	R	Spindle Speed Override (%) selected by program.
SDNCSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
SPLCSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
SCNCSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SSLIMI	R	R	R	Spindle speed limit, in rpm, active at the CNC.
SDNCSL	R	R	R/W	Spindle speed limit selected via DNC.
SPLCSL	R	R/W	R	Spindle speed limit selected via PLC.
SPRGSL	R	R	R	Spindle speed limit selected by program.
SPOSS	R	R	R	Spindle real position, (max ±999999999 in 0.0001º)
SRPOSS	R	R	R	Spindle real position, (between 0º and 360º in 0.0001º)
STPOSS	R	R	R	Spindle theoretical position, (max ±999999999 in 0.0001º)
SRTPOS	R	R	R	Spindle theoretical position (between 0º and 360º in 0.0001º)
SFLWES	R	R	R	Spindle following error when in Closed loop (M19), in degrees

VARIABLES ASSOCIATED WITH THE LIVE TOOL *(Section 11.2.11)*

Variable	CNC	PLC	DNC	
ASPROG	R	-	-	Rpm programmed in M45 S (within the associated subroutine)
LIVRPM	R	R	-	Rpm of the live tool in TC mode.

VARIABLES ASSOCIATED WITH THE PLC

(Section 11.2.12)

Variable	CNC	PLC	DNC	
PLCMSG	R	-	R	Number of the active PLC message with the highest priority.
PLCIn	R/W	-	-	32 PLC inputs starting from (n).
PLCOn	R/W	-	-	32 PLC outputs starting from (n).
PLCMn	R/W	-	-	32 PLC marks starting from (n).
PLCRn	R/W	-	-	Indicated (n) Register.
PLCTn	R/W	-	-	Indicated (n) Timer's count.
PLCCn	R/W	-	-	Indicated (n) Counter's count.

VARIABLES ASSOCIATED WITH GLOBAL AND LOCAL ARITHMETIC PARAMETERS

(Section 11.2.13)

Variable	CNC	PLC	DNC	
GUP n	-	R/W	-	Global parameter (n) (100-P299).
LUP (a,b)	-	R/W	-	Local parameter (b) and its nesting level (a). (P0-P25).
CALLP	R	-	-	Indicates which local parameters have been defined by means of a PCALL or MCALL instruction (calling a subroutine).

SERCOS VARIABLES

(Section 11.2.14)

Variable	CNC	PLC	DNC	
SETGE(X-C)	W	W	-	Work gear and parameter set for (X-C) axis drive
SETGES	W	W	-	Work gear and parameter set for main spindle drive
SSETGS	W	W	-	Work gear and parameter set for 2nd spindle drive
SVAR(X-C) id	R/W	-	-	Sercos variable for (X-C) axis identifier "id"
SVARS id	R/W	-	-	Sercos variable for main spindle identifier "id"
SSVAR id	R/W	-	-	Sercos variable for 2nd spindle identifier "id"
TSVAR(X-C) id	R	-	-	Third attribute of the sercos variable of (X-C) axis identifier "id"
TSVARS id	R	-	-	Third attribute of the sercos variable of main spindle identifier "id"
TSSVAR id	R	-	-	Third attribute of the sercos variable of 2nd spindle identifier "id"

OTHER VARIABLES

(Section 11.2.15)

Variable	CNC	PLC	DNC	
OPMODE	R	R	R	Operating mode.
OPMODA	R	R	R	Operating mode when working in the main channel.
OPMODB	R	R	R	Type of simulation.
OPMODC	R	R	R	Axes selected by handwheel.
NBTOOL	R	R	R	Number of the tool being managed
PRGN	R	R	R	Number of the program in execution.
BLKN	R	R	R	Label number of the last executed block.
GSn	R	-	-	Status of the indicated G function (n).
GGSA	-	R	R	Status of functions G00 thru G24.
GGSB	-	R	R	Status of functions G25 thru G49.
GGSC	-	R	R	Status of functions G50 thru G74.
GGSD	-	R	R	Status of functions G75 thru G99.
MSn	R	-	-	Status of the indicated M function (n)
GMS	-	-	R	Status of M functions: M (0..6, 8, 9, 19, 30, 41..44)
PLANE	R	R	R	Axes which form the active main plane.
MIRROR	R	R	R	Active mirror images.
SCALE	R	R	R	Active general Slaving factor.
SCALE(X-C)		R	R	R Scaling Factor applied only to the indicated axis.
PRBST	R	R	R	Returns the status of the probe.
CLOCK	R	R	R	System clock in seconds.
TIME	R	R	R/W	Time in Hours, minutes and seconds.
DATE	R	R	R/W	Date in Year-Month-Day format
TIMER	R/W	R/W	R/W	Clock activated by PLC, in seconds.
CYTIME	R	R	R	Time to execute a part in hundredths of a second.
PARTC	R/W	R/W	R/W	Part counter of the CNC.
FIRST	R	R	R	Flag to indicate first time of program execution.
KEY	R/W*	R/W	R/W	keystroke code.
KEYSRC	R/W	R/W	R/W	Keystroke source, 0=keyboard, 1=PLC, 2=DNC
ANAIn	R	R	R	Voltage (in volts) of the indicated analog input (n).
ANAOn	R/W	R/W	R/W	Voltage (in volts) to apply to the indicated output (n).
CNCERR	-	R	R	Active CNC error number.
PLCERR	-	-	R	Active PLC error number.
DNCERR	-	R	-	Number of the error generated during DNC communications.
AXICOM	R	R	R	Pair of axes toggled with function G28
TANGAN	R	R	R	Associated with G45. Angular position, in º, referred to the path.

Warning

The "Key" variable at the CNC may be written only via the user channel.

HIGH LEVEL PROGRAMMING

DISPLAY STATEMENTS

(Section 12.2)

(ERROR whole number, "error text")	Stops execution of program and displays indicated error.
(MSG "message")	Displays indicated message.
(DGWZ expression 1, ... expression 4)	Define the graphics display area

ENABLING / DISABLING STATEMENTS

(Section 12.3)

(ESBLK and DSBLK)	The CNC executes all the blocks which are found between ESBLK and DSBLK as if they were a single block.
(ESTOP and DSTOP)	Enable (ESTOP) and disable (DSTOP) of the Stop key and the external Stop signal (PLC)
(EFHOLD and DFHOLD)	Enable (EFHOLD) and disable (DFHOLD) of the Feed-Hold input (PLC)

FLOW CONTROLLING STATEMENTS

(Section 12.4)

(GOTO N(expression))	Causes a jump within the same program, to the block defined by label N(expression)
(RPT N(expression), N(expression))	Repeats the execution of the part of a program existing between two blocks defined by means of labels N(expression)
(IF condition <action1> ELSE <action2>)	Analyzes the given condition which must be a comparative expression. If the condition is true (result equals 1), <action1> will be executed, otherwise (result equals 0) <action2> will be executed.

SUBROUTINE STATEMENTS

(Section 12.5)

(SUB integer)	Definition of subroutine
(RET)	End of subroutine
(CALL (expression))	Call to subroutine
(PCALL (expression, (assignment statement), (assignment statement),...)	Call to a subroutine. Besides, allows the initialization, by means of assignment mnemonics, of up to 26 local parameters of this subroutine.
(MCALL (expression), (assignment statement), (assignment statement),...)	The same as PCALL, but converting the subroutine indicated into a modal subroutine.
(MDOFF)	Cancellation of modal subroutine
(PROBE (expression),(assignment statement), (assignment statement),...)	Executes a probing canned cycle, its parameters being initialized by means of assignment statements.
(REPOS X, Y, Z,...)	It must always be used inside interruption subroutines and it facilitates the repositioning of the machine axes to the interruption point.

PROGRAM STATEMENTS

(Section 12.6)

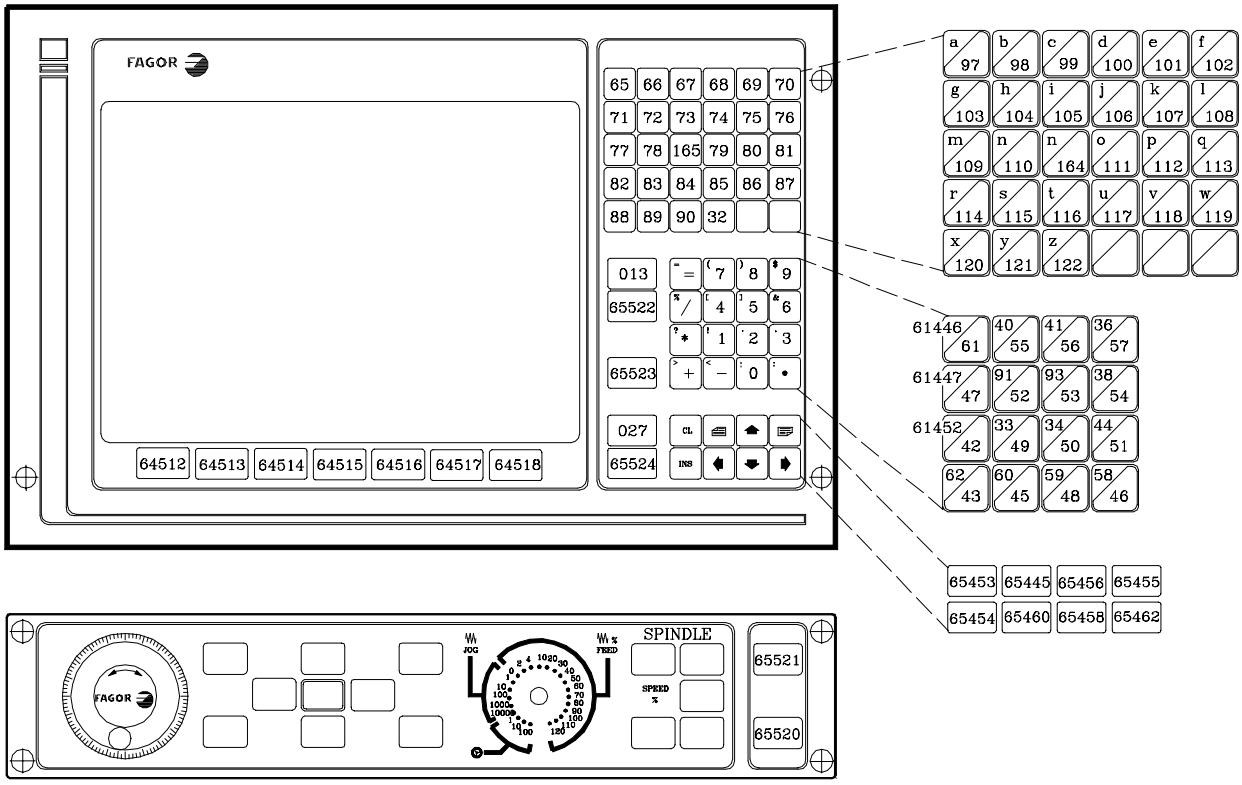
(EXECP(expression), (directory)	Starts the execution of the program
(OPEN P(expression), (target directory), A/D, "program comment")	Starts generating a new program and allows it to be associated with a program comment.
(WRITE <block text>)	Adds the information contained in <block text> after the last program block of the program which was being generated with OPEN P, as a new program block.

CUSTOMIZING STATEMENTS

(Section 12.7)

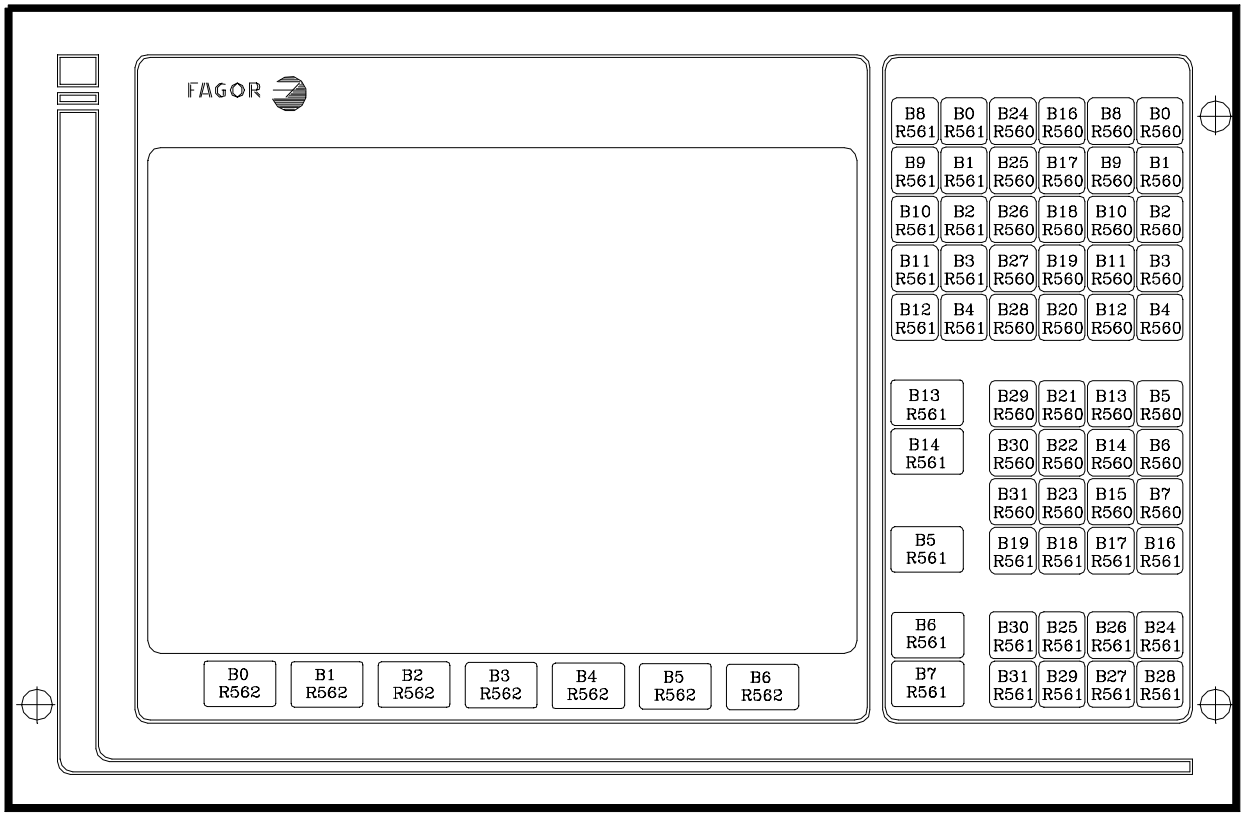
(PAGE(expression))	Displays the user page number (0-255) or system page number (>1000) indicated.
(SYMBOL (expression 1),(expression 2),(expression 3)	Displays the symbol (0-255) indicated by expression 1 Its position on the screen is defined by expression 2 (row,0-639) and by expression 3 (column,0-335).
(IB(expression)=INPUT"text",format)	Displays the text indicated in the data input window and stores the data input by the user in the input variable (IBn).
(ODW(expression 1), (expression 2), (expression 3)	Defines and draws a white window on screen (1 row x 14 columns). Its position on screen is defined by expression 2(row) and by expression 3 (column).
(DW (expression 1)=(expression 2), DW(expression 3) = (expression 4),...)	Displays the numerical data indicated by expression 2,4,.. in windows indicated by the value of expression 1,3....
(SK (expression 1)="text 1", (expression 2)="text 2",...)	Defines and displays the new softkey menu indicated.
(WKEY)	Stops the execution of a program until a key is pressed.
(WBUF"text"(expression))	Adds the text and the value of the expression, once this has been evaluated, to the block which is being edited and in the data input window.
(SYSTEM)	Ends the execution of user customized program and returns to standard CNC menu.

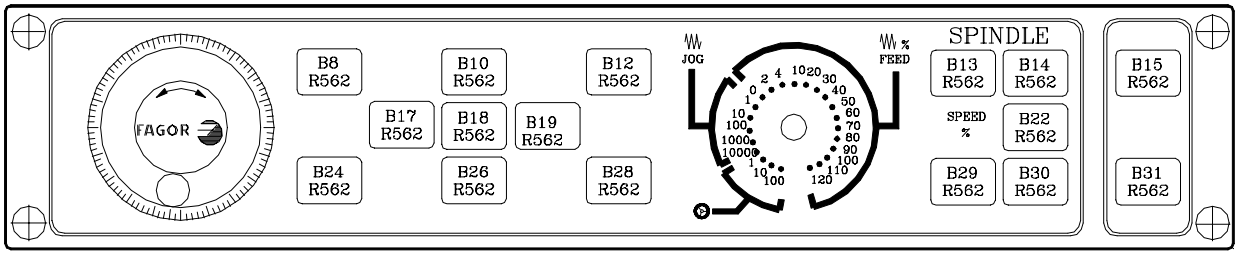
KEY CODES


FAGOR
64512 64513 64514 64515 64516 64517 64518
65 66 67 68 69 70
71 72 73 74 75 76
77 78 165 79 80 81
82 83 84 85 86 87
88 89 90 32
013
65522
65523
027
65524
a 97 b 98 c 99 d 100 e 101 f 102
g 103 h 104 i 105 j 106 k 107 l 108
m 109 n 110 n 164 o 111 p 112 q 113
r 114 s 115 t 116 u 117 v 118 w 119
x 120 y 121 z 122
61446 61 40 55 41 56 36 57
61447 47 91 52 93 53 38 54
61452 42 33 49 34 50 44 51
62 43 60 45 59 48 58 46
65453 65445 65456 65455
65454 65460 65458 65462

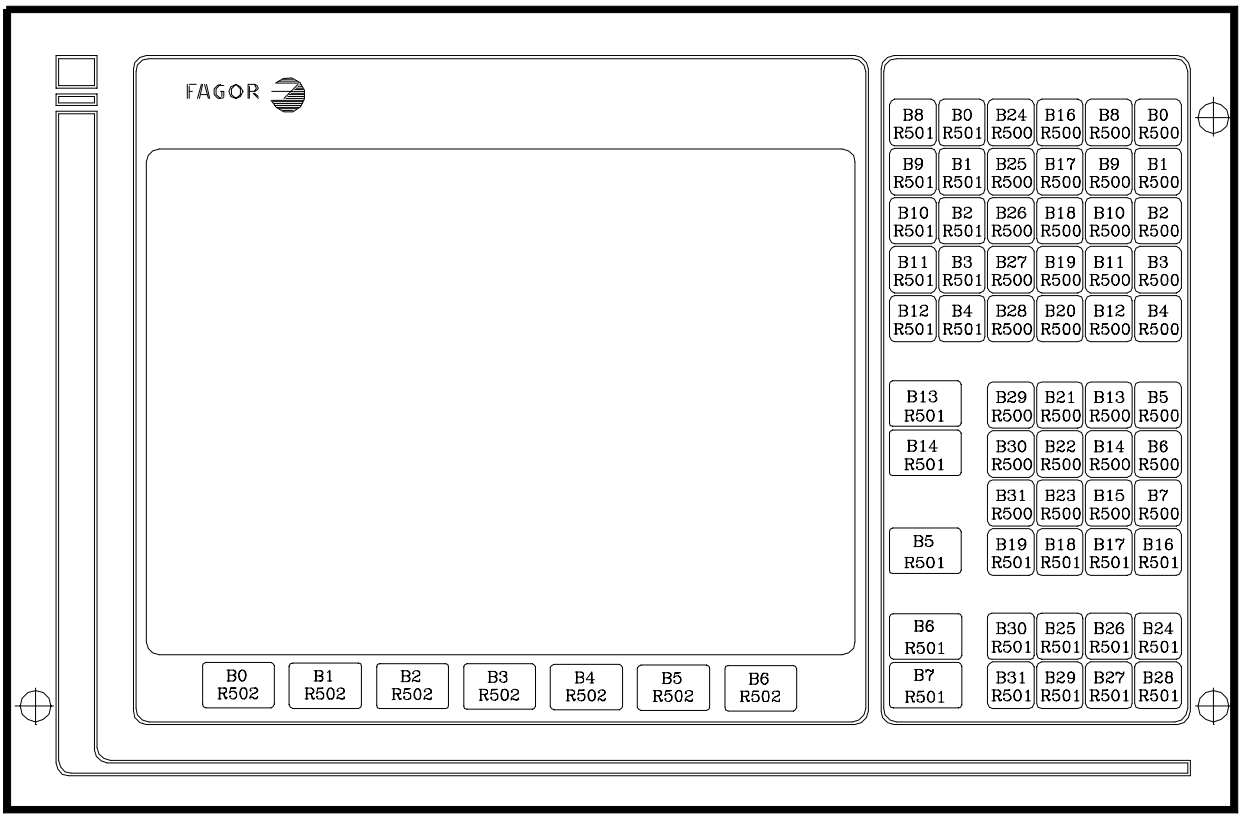
FAGOR
JOG
FEED
SPINDLE
SPEED %
65521
65520

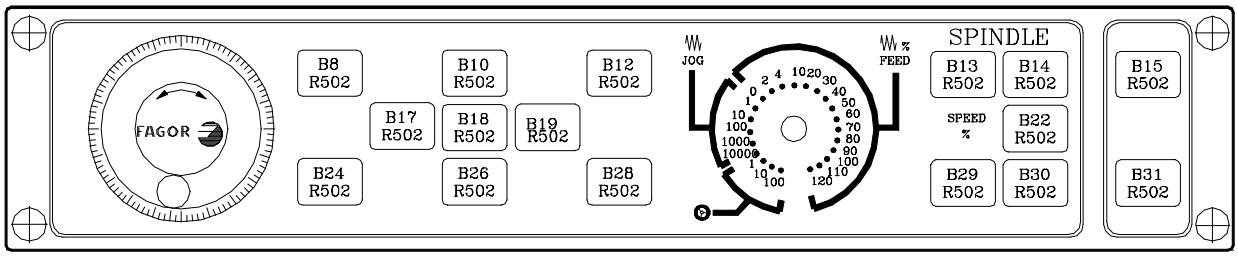
LOGIC OUTPUTS FOR KEY CODE STATUS





KEYS INHIBITING CODES





G PROGRAMMING ASSISTANCE SYSTEM PAGES

These pages can be displayed by means of the high level mnemonic “PAGE”. They all belong to the CNC system and are used as help pages for their respective functions.

GLOSSARY HELP

Page 1000 Preparatory functions G00-G09.
Page 1001 Preparatory functions G10-G19.
Page 1002 Preparatory functions G20-G44.
Page 1003 Preparatory functions G53-G59.
Page 1004 Preparatory functions G60-G69.
Page 1005 Preparatory functions G70-079.
Page 1006 Preparatory functions G80-G89.
Page 1007 Preparatory functions G90-G99.
Page 1008 Miscellaneous (auxiliary) functions M.
Page 1009 Miscellaneous M functions with the symbol for next page.
Page 1010 Coincides with 250 of the directory if it exists.
Page 1011 Coincides with 251 of the directory if it exists.
Page 1012 Coincides with 252 of the directory if it exists.
Page 1013 Coincides with 253 of the directory if it exists.
Page 1014 Coincides with 254 of the directory if it exists.
Page 1015 Coincides with 255 of the directory if it exists.
Page 1016 High level language listing (from A to G)
Page 1017 High level language listing (from H to N)
Page 1018 High level language listing (from 0 to S)
Page 1019 High level language listing (from T to Z)
Page 1020 High level accessible variables (1st part)
Page 1021 High level accessible variables (2nd part)
Page 1022 High level accessible variables (3rd part)
Page 1023 High level accessible variables (4th part)
Page 1024 High level accessible variables (5th part)
Page 1025 High level accessible variables (6th part)
Page 1026 High level accessible variables (7th part)
Page 1027 High level accessible variables (8th part)
Page 1028 High level accessible variables (9th part)
Page 1029 High level accessible variables (10th part)
Page 1030 High level accessible variables (11th part).
Page 1031 High level accessible variables (12th part).

SYNTAX ASSISTANCE: ISO LANGUAGE

Page 1033 Program block structure
Page 1034 Positioning and linear interpolation: G00,G01 (1st part)
Page 1035 Positioning and linear interpolation: G00,G01 (2nd part)
Page 1036 Circular interpolation: G02, G03 (1st part)
Page 1037 Circular interpolation: G02, G03 (2nd part)
Page 1038 Circular-helical interpolation: G02, G03
Page 1039 Arc tangent to previous path: G08 (1st part)
Page 1040 Arc tangent to provious path: G08 (2nd part)
Page 1041 Arc defined by three points: G09 (1st part)
Page 1042 Arc defined by three points: G09 (2nd part)
Page 1043 Threadcutting: G33
Page 1044 Controlled corner rounding: G36
Page 1045 Tangential entry: G37
Page 1046 Tangential exit: G38
Page 1047 Chamfer blend: G39
Page 1048 Dwell/Block preparation stop: G04, G04K.
Page 1049 Round/Square corner: G07, G05.
Page 1050 Mirror image: G11, G12, G13, G14.
Page 1051 Planes and longitudinal axis selection: G16,G17, G18, G19, G15.
Page 1052 Work zones: G21, G22.
Page 1053 Tool radius compensation: G40,G41,G42.
Page 1054 Zero offsets.
Page 1055 Millimeters/inches: G71, G70.
Page 1056 Scaling factor: G72.
Page 1057 Machine reference search: G74
Page 1058 Probing: G75.
Page 1059 Slaved axis: G77, G78.
Page 1060 Absolute/incremental programming: G90, G91.
Page 1061 Coordinate and polar origin preset: G92,G93.
Page 1062 Feedrate programming: G94,G95.
Page 1063 Auxiliary function programming F,S,T and D.
Page 1064 Auxiliary function M programming.

SYNTAX ASSISTANCE: CNC TABLES

Page 1090 Tool Offset table.
Page 1091 Tool table
Page 1092 Tool magazine table.
Page 1093 Miscellaneous (auxiliary) function M table.
Page 1094 Zero offset table.
Page 1095 Leadscrew error compensation tables.
Page 1096 Crossed compensation table.
Page 1097 Machine parameter tables.
Page 1098 User parameter tables.
Page 1099 Password table.

SYNTAX ASSISTANCE: HIGH LEVEL

Page 1100 : ERROR and MSG mnemonics.
Page 1101 : GOTO and RPT mnemonics.
Page 1102 : OPEN and WRITE mnemonics.
Page 1103 : SUB and RET mnemonics.
Page 1104 : CALL, PCALL, MCALL, MDOFF and PROBE mnemonics.
Page 1105 : DSBLK, ESBLK, DSTOP, ESTOP, DFHOLD, EFHOLD mnemonics.
Page 1106 : IF mnemonic.
Page 1107 : Assignment blocks.
Page 1108 : Mathematical expressions.
Page 1109 : PAGE mnemonic.
Page 1110 : ODW mnemonic.
Page 1111 : DW mnemonic.
Page 1112 : IB mnemonic.
Page 1113 : SK mnemonic.
Page 1114 : WKEY and SYSTEM mnemonics.
Page 1115 : KEYSRC mnemonic.
Page 1116 : WBUF mnemonic.
Page 1117 : SYMBOL mnemonic.

SYNTAX ASSISTANCE: CANNED CYCLES

Page 1076 : Pattern Repeat canned cycle: G66.
Page 1078 : Stock removal along X canned cycle: G68.
Page 1079 : Stock removal along Z canned cycle: G69.
Page 1080 : Turning canned cycle with straight sections: G81.
Page 1081 : Facing canned cycle with straight sections: G82.
Page 1082 : Drilling canned cycle: G83.
Page 1083 : Turning canned cycle with circular sections: G84.
Page 1084 : Facing canned cycle with circular sections: G85.
Page 1085 : Longitudinal threadcutting canned cycle: G86.
Page 1086 : Face threadcutting canned cycle: G87.
Page 1087 : Grooving canned cycle along X axis: G88.
Page 1088 : Grooving canned cycle along Z axis: G89.
Page 1089 : Tool geometry.

H MAINTENANCE

Cleaning:

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the CNC and, consequently, possible malfunctions.

On the other hand, accumulated dirt can sometimes act as an electrical conductor and shortcircuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol.

Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to :

1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

Avoid the action of solvents such as Chlorine hydrocarbons , Benzole , Esters and Ether which can damage the plastics used to make the unit's front panel.

Preventive Inspection:

If the CNC does not turn on when actuating the start-up switch, verify that the monitor fuses are in good condition and that they are the right ones.

To check the fuses, first disconnect the power to the CNC.

Do not open this unit.
Only personnel authorized by Fagor Automation may open this module.

Do not handle the connectors with the unit connected to main AC power.
Before handling these connectors, make sure that the unit is not connected to main AC power.

Note :
Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.

List of materials, parts that could be replaced

		Code
Central Unit	3 modules 6 modules	83060100 83060101
CPU module	Mill Lathe Sercos board	83090122 83090123 83160110
Axes module		83150100
I/O module		83210100
I/O Tracing module		83220100
Sercos module		83160100
Cover (empty module)		83300100
CPU Turbo		80500077

9" Amber monitor (no keyboard)	MC & TC	83390002
9" Amber monitor (with keyboard)	Mill Lathe	83390000 83390001
10" Color monitor (no keyboard)	MC & TC	83390004
10" Color monitor (with keyboard)	Mill Lathe	83420001 83420003
11" LCD monitor (no keyboard)	MC & TC	83480100
11" LCD Monitor (with keyboard)	Mill Lathe M & MC T & TC	83480101 83480102 83480103 83480104
14" Color monitor (no keyboard)		83390003
14" Color monitor (with keyboard)		83420004

		Código
Operator panel (no handwheel)	Mill Lathe	80300010 80300011
Operator panel (with handwheel)	Mill Lathe	80300014 80300015
Operator panel	MC TC	83540020 83540002
switcher board		83900000

Video cables	5m 10m 15m 20m 25m	83540020 83630021 83630022 83630023 83630024
Keyboard cables	2m 5m 10m 15m 20m 25m	83630010 83630004 83630005 83630006 83630008 83630026

Configuration card MemKey Card	4 Mb 8 Mb 16 Mb 24 Mb	83120150 83120160 83120161 83120162

Video adapter (digital - analog)		8C401001
Video duplicator	(DVD)	83900001
DNC software		80500115

Available manuals

Mill Model		Standard software (code)	Advanced software (code)
OEM Manuals	Spanish English French German Italian portuguese	03753400 03753401	03753460 03753461
User Manuals	Spanish English French German Italian portuguese	03753410 03753411 03753412 03753413 03753414 03753415	03753470 03753471 03753472 03753473 03753474 03753475
Conversational model (MC)	Spanish English French German Italian portuguese	03753440 03753441 03753442 03753443 03753444 03753445	03753500 03753501 03753502 03753503 03753504 03753505

Lathe Model		Standard software (code)	Advanced software (code)
OEM Manuals	Spanish English French German Italian portuguese	03753420 03753421	03753480 03753481
User Manuals	Spanish English French German Italian portuguese	03753430 03753431 03753432 03753433 03753434 03753435	03753490 03753491 03753492 03753493 03753494 03753495
Conversational model (TC)	Spanish English French German Italian portuguese	03753450 03753451 03753452 03753453 03753454 03753455	03753510 03753511 03753512 03753511 03753514 03753515

8055T CNC

ERROR TROUBLESHOOTING MANUAL

Ref. 9905 (ing)

INDEX

Programming errors **1**
(0001-0255)

Preparation and execution errors **29**
(1000-1238)

Hardware errors **45**
(2000-2028)

PLC errors **48**
(3000-3004)

Drive errors **49**
(4000-4025)

Table data errors **51**

Errors in 8055TC operating mode **54**

Alphabetical index **65**

PROGRAMMING ERRORS

0001 'Empty line.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When trying to enter into a program or execute an empty block or containing the label (block number).

2.- In the «Pattern repeat canned cycle (G66)», «Roughing canned cycle along the X axis (G68)» or Roughing canned cycle along the Z axis (G69)». Parameter "S" (beginning of the profile) is greater than parameter "E" (end of profile).

SOLUTION The solution for each cause is:

1.- The CNC cannot enter into the program or execute an empty line. To do that, use the «;» symbol at the beginning of that block. The CNC will ignore the rest of the block.

2.- The value of parameter "S" (block where the profile definition begins) must be lower than the value of parameter "E" (block where the profile definition ends).

0002 'Improper data'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When editing an axis coordinate after the cutting conditions (F, S, T or D) or the «M» functions.

2.- When the marks of the block skip (conditional block /1, /2 or /3) are not at the beginning of the block.

3.- When programming a block number greater than 9999 while programming in ISO code.

4.- While programming in high-level, the value of the RPT instruction exceeds 9999.

SOLUTION The solution for each cause is:

1/2.- Remember that the programming order is:

1.- Block skip (conditional block /1, /2 or /3).
2.- Label (N).
3.- «G» functions.
4.- Axes coordinates (X, Y, Z…).
5.- Machining conditions (F, S, T, D).
6.- «M» functions.

All the data need not be programmed.

3.- Correct the block syntax. Program the labels between 0 and 9999

4.- Correct the block syntax. Program the labels between 0 and 9999

0003 'Improper data order.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The machining conditions or the tool data have been programmed in the wrong order.

SOLUTION Remember that the programming order is:

… F— S— T— D— …

All the data need not be programmed.

0004 'No more information allowed in the block.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When editing a «G» function after an axis coordinate.

2.- When trying to edit some data after a «G» function (or after its associated parameters) which must go alone in the block (or which only admits its own associated data).

3.- When assigning a numeric value to a parameter that does not need it.

SOLUTION The solution for each cause is:

1.- Remember that the programming order is:

1.- Block skip (conditional block /1, /2 or /3).
2.- Label (N).
3.- «G» functions.
4.- Axes coordinates. (X, Y, Z…).
5.- Machining conditions (F, S, T, D).
6.- «M» functions.

All the data need not be programmed.

2.- There are some «G» functions which carry associated data in the block. Maybe, this type of functions do not let program other type of information after their associated parameters. On the other hand, neither machining conditions, (F, S), tool data (T, D) nor «M» functions may be programmed.

3.- There are some «G» functions having certain parameters associated to them which do not need to be defined with values.

0005 'Repeated information'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The same data has been entered twice in a block.

SOLUTION Correct the syntax of the block. The same data cannot be defined twice in a block.

0006 'Improper data format'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While defining the parameters of a machining canned cycle, a negative value has been assigned to a parameter which only admits positive values.

SOLUTION Verify the format of the canned cycle. In some canned cycles, there are parameters which only accept positive values.

0007 'Incompatible G functions.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When programming in the same block two «G» functions which are incompatible with each other.

2.- When trying to define a canned cycle in a block containing a nonlinear movement (G02, G03, G08, G09, G33).

SOLUTION The solution for each cause is:

1.- There are groups of «G» functions which cannot go together in the block because they involve actions incompatible with each other. For example:

G01/G02: Linear and circular interpolation
G41/G42: Left-hand or right-hand tool radius compensation.

This type of functions must be programmed in different blocks.

2.- A canned cycle must be defined in a block containing a linear movement. In other words, to define a cycle, a "G00" or a "G01" must be active. Nonlinear movements (G02, G03, G08 and G09) may be defined in the blocks following the profile definition.

0008 'Nonexistent G function'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A nonexistent «G» function has been programmed.

SOLUTION Check the syntax of the block and verify that a different «G» function is not being edited by mistake.

0009 'No more G functions allowed in the block'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A «G» function has been programmed after the machining conditions or after the tool data.

SOLUTION Remember that the programming order is:

1.- Block skip (conditional block /1, /2 or /3).
2.- Label (N).
3.- «G» functions.
4.- Axes coordinates. (X, Y, Z…).
5.- Machining conditions (F, S, T, D).
6.- «M» functions.

All the data need not be programmed.

0010 'No more M functions allowed in the block'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE More than 7 «M» functions have been programmed in a block.

SOLUTION The CNC does not let program more than 7 «M» functions in a block. To do so, write them in a separate block. The «M» functions may go alone in a block.

0011 'This G or M function must be alone.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The block contains either a «G» or an «M» function that must go alone in the block.

SOLUTION Write it alone in the block.

0012 'Program F, S, T, D before the M functions.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A machining condition (F, S) or tool data (T, D) has been programmed after the «M» functions.

SOLUTION Remember that the programming order is:

… F— S— T— D— M—

Up to 7 «M» functions may be programmed .
All the data need not be programmed.

0014 'Do not program labels by parameters.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A label (block number) has been defined with a parameter.

SOLUTION The programming of a block number is optional, but it cannot be defined with a parameter, only with a number between 0 and 9999.

0015 'Number of repetitions not possible.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A repetition has been programmed wrong or the block does not admit repetitions.

SOLUTION High level instructions do not admit a number of repetitions at the end of the block. To do a repetition, assign to the block to be repeated a label (block number) and use the RPT instruction.

0017 'Program: G16 axis-axis.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function «Main plane selection by two axes (G16)» one of the two parameters for the axes has not been programmed.

SOLUTION Check the syntax of the block. The definition of the "G16" function requires the name of the axes defining the new work plane.

0018 'Program: G22 K(1/2/3/4) S(0/1/2).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function «Enable/Disable work zones (G22)» the type of enable or disable of the work zone has not been defined or it has been assigned the wrong value.

SOLUTION The parameter for enabling or disabling the work zones "S" must always be programmed and it may take the following values.

- S=0: The work zone is disabled.
- S=1: It is enabled as a no-entry zone.
- S=2: It is enabled as a no-exit zone.

0019 'Program: work zone K1, K2, K3 or K4.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- A "G20", "G21" or "G22" function has been programmed without defining the work zone K1, K2, K3 or K4.

2.- The programmed work zone is smaller than 0 or greater than 4.

SOLUTION The solution for each cause is:

1.- The programming format for functions "G20", "G21" and "G22" is:

G20 K— X...C±5.5 (Definition of lower work zone limits).
G21 K— X...C±5.5 (Definition of upper work zone limits).
G22 K— S— (Enable/disable work zones).

Where:

- K : Is the work zone.
- X...C : Are the axes where the limits are defined.
- S : Is the type of work zone enable.

2.- The "K" work zone may only have the values of K1, K2, K3 or K4.

0020 'Program G36-G39 with R+5.5.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "G36" or "G39" function, the "R" parameter has not been programmed or it has been assigned a negative value.

SOLUTION To define "G36" or "G39", parameter "R" must also be defined and with a positive value).

G36: R= Rounding radius.
G39: R= Distance between the end of the programmed path and the point to be chamfered.

0021 'Program: G72 S5.5 or axes.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When programming a general scaling factor (G72) without the scaling factor to apply.

2.- When programming a particular scaling factor (G72) to several axes, but the axes have been defined in the wrong order.

SOLUTION Remember that this function must be programmed in the following order:

- "G72 S5.5" When applying a general scaling factor (to all axes).
- "G72 X…C5.5" When applying a particular scaling factor to one or several axes.

0023 'Block incompatible when defining a profile.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the set of blocks defining a profile, there is a block containing a «G» function that cannot belong to the profile definition.

SOLUTION The «G» functions available in the profile definition are:

G00: Beginning of the profile.
G01: Linear interpolation.
G02/G03: Clockwise/counterclockwise interpolation.
G06: Circle center in absolute coordinates.
G08: Arc tangent to previous path.
G09: Three point arc.
G36: Controlled corner rounding
G39: Chamfer.
G53: Programming with respect to home.
G70/G71: Inch/metric programming.
G90/G91: Programming in absolute/incremental coordinates.
G93: Polar origin preset.

0024 'High level blocks not allowed when defining a profile.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the set of blocks defining a profile, a high level block has been programmed.

SOLUTION The profile must be defined in ISO code. No high level instructions are allowed (GOTO, MSG, RPT ...).

0025 'Program: G77 axes (2 thru 6).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Axis slaving (G77)» function, the parameters for the axes have not been programmed.

SOLUTION The programming of "G77" function requires at least two axes.

0026 'Program: G93 I J.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Polar origin preset (G93)» function, some of the parameters for the new polar origin have not been programmed.

SOLUTION Remember that the programming format for this function is:

G93 I— J—

The "I", "J" values are optional, but if programmed, both must be programmed and they indicate the new polar origin.

0028 'G2 or G3 not allowed when programming a canned cycle.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A canned cycle has been attempted to execute while the "G02", "G03" or "G33" functions were active.

SOLUTION To execute a canned cycle, "G00" or "G01" must be active. Maybe, a "G02" or "G03" function was activated in the M code history instead. Check that these functions are not active when the canned cycle is defined.

0029 'G84-85: X Z Q R C [D L M F H] I K.'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Turning canned cycle with arcs (G84)» or «Facing canned cycle with arcs (G85)» have been programmed wrong. The probable causes might be:

1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:

X-Z : Profile starting point.
Q-R : Profile end point.
C : Cutting pass.
I-K : Distance from the starting point to the center of the arc.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0030 'G86-87: X Z Q R I B [D L] C [J A].'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Longitudinal threading canned cycle (G86)» or «Face threading canned cycle (G87)» have been programmed wrong. The probable causes might be:

1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:

X-Z : Starting point of the thread.
Q-R: End point of the thread.
I : Depth of the thread.
B : Cutting pass.
C : Thread pitch

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0031 'G88-G98: X Z Q R [C D K].'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Grooving canned cycle along the X axis (G88)» or «Grooving canned cycle along the Z axis (G89)» have been programmed wrong. The probable causes might be:

1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:

X-Z : Starting point of the groove.
Q-R: End point of the groove.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0032 'G66: X Z I C [A L M H] S E.'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Pattern repeat canned cycle (G66)» have been programmed wrong. The probable causes might be:
1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:
X-Z : Profile starting point.
I : Residual stock.
C : Cutting pass.
S : Block where the profile geometry description begins.
E : Block where the profile geometry description ends.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0033 'G68-G69: X Z C [D L M F H] S E .'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Roughing canned cycle along the X axis (G68)» or «Roughing canned cycle along the Z axis (G69)» have been programmed wrong. The probable causes might be:
1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:
X-Z : Profile starting point.
C : Cutting pass.
S : Block where the profile geometry description begins.
E : Block where the profile geometry description ends.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0034 'G81-G82: X Z Q R C [D L M F H].'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Turning canned cycle with straight sections (G81)» or «Facing canned cycle with straight sections (G82)» have been programmed wrong. The probable causes might be:
1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:
X-Z : Profile starting point.
Q-R: Profile end point.
C : Cutting pass.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0035 'G83: X Z I B [D K H C].'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Axial drilling / tapping canned cycle (G83)» have been programmed wrong. The probable causes might be:
1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:
X-Z : Position of the machining operation.
I : Depth of the machining operation.
B : Type of operation to be carried out.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0036 'G60-G61: X Z I B Q A J [D K H C] S.'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Face drilling / tapping canned cycle (G60)» or «Longitudinal drilling / tapping canned cycle (G61)» have been programmed wrong. The probable causes might be:

1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.
3.- A parameter has been programmed which does not match the calling format.

SOLUTION This type of machining requires the programming of the following parameters:

X-Z : Position of the machining operation.
I : Depth of the machining operation.
B : Type of operation to be carried out.
Q : Angular positioning of the first machining operation.
A : Angular step between machining operations.
J : Number of machining operations.
S : Speed and turning direction of the live tool.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0037 'G62-G63: X Z L I Q A J [D] F S.'

DETECTED While editing or executing programs via DNC.

CAUSE The parameters of the «Axial slot milling canned cycle (G62)» or «Radial slot milling canned cycle (G62)» have been programmed wrong. The probable causes might be:

1.- A mandatory parameter is missing.
2.- The cycle parameters are programmed in the wrong order.

SOLUTION This type of machining requires the programming of the following parameters:

X-Z : Position of the slot.
L : Length of the slot.
I : Depth of the slot.
Q : Angular position of the first slot.
A : Angular step between slot.
J : Number of slots.
F : Feedrate.
S : Speed and turning direction of the live tool.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0043 'Incomplete Coordinates.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- During simulation or execution, when trying to make a movement defined with only one coordinate of the end point or without defining the arc radius while a «circular interpolation (G02/G03) is active.
2.- During editing, when editing a circular movement (G02/G03) by defining only one coordinate of the end point or not defining the arc radius.

SOLUTION The solution for each cause is:

1.- A "G02" or "G03" function may be programmed previously in the program history. In this case, to make a move, both coordinates of the end point and the arc radius must be defined. To make a linear movement, program "G01".
2.- To make a circular movement (G02/G03), both coordinates of the end point and the arc radius must be programmed.

0045 'Polar coordinates not allowed.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE When «Programming with respect to home (G53)», the end point has been defined in polar or cylindrical coordinates or in Cartesian coordinates with an angle.

SOLUTION When programming with respect to home, only Cartesian coordinates may be programmed.

0046 'Axis does not exist.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A block has been edited whose execution implied the movement of a nonexistent axis.

SOLUTION Check that the name of the axis is correct.

0047 'Program axes.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE No axis has been programmed in a function requiring an axis.

SOLUTION Some instructions require the programming of axes (REPOS, G14, G20, G21…).

0048 'Incorrect order of axes.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The axis coordinates have not been programmed in the correct order or an axis has been programmed twice in the same block.

SOLUTION Remember that the correct programming order for the axes is:

X— Y— Z— U— V— W— A— B— C—

All axes need not be programmed:

0049 'Point incompatible with active plane.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When trying to do a circular interpolation, the end point is not in the active plane.

2.- When trying to do a tangential exit in a path that is not in the active plane.

SOLUTION The solution for each cause is:

1.- Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, circular interpolations can only be carried out on the main axes defining that plane. To define a circular interpolation in another plane, it must be defined beforehand.

2.- Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.

0053 'Program pitch.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Electronic threading cycle (G33)» the parameter for the thread pitch is missing.

SOLUTION Remember that the programming format for this function is:

G33 X...C— L—

Where:

L : Is the thread pitch.

0054 'Pitch programmed incorrectly.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A helical interpolation has been programmed with the wrong or negative pitch.

SOLUTION Remember that the programming format is:

G02/G03 X— Y— I— J— Z— K—

Where:

K : is the helical pitch (always positive value).

0057 'Do not program a slaved axis.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The various causes might be:

1.- When trying to move an axis alone while being slaved to another one.

2.- When trying to slave an axis that is already slaved using the G77 function «Electronic axis slaving».

SOLUTION The solution for each cause is:

1.- A slaved axis cannot be moved separately. To move a slaved axis, its master axis must be moved. Both axes will move at the same time.

Example: If the Y axis is slaved to the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).

To unslave the axis, program "G78".

2.- An axis cannot be slaved to two different axes at the same time. To unslave the axes, program "G78".

0058 'Do not program a GANTRY axis.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When trying to move an axis alone while being slaved to another one as a GANTRY axis

2.- When defining an operation on a GANTRY axis. (Definition of work zone limits, planes, etc.).

SOLUTION The solution for each cause is:

1.- A GANTRY axis cannot be moved separately. To move a GANTRY axis, its associated axis must be moved. Both axes will move at the same time.

Example: If the Y axis is a GANTRY axis associated with the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).

GANTRY axes are defined by machine parameter.

2.- The axes defined as GANTRY cannot be used in the definition of operations or movements. These operations are defined with the main axis that the GANTRY axis is associated with.

0059 'HIRTH axis: program only integer values.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A rotation of a HIRTH axis has been programmed with a decimal value.

SOLUTION HIRTH axes do not accept decimal angular values. They must be full degrees.

0061 'ELSE not associated with IF.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- While editing in High level language, when editing the "ELSE" instruction without having previously programmed an "IF".

2.- When programming in high level language, an "IF" is programmed without associating it with any action after the condition.

SOLUTION Remember that the programming formats for this instruction are:

(IF *(condition) <action1>)*

(IF *(condition)* <*action1*> ELSE <*action2*>)

If the *condition* is true, it executes the <*action1*>, otherwise, it executes the <*action2*>.

0062 'Program label N(0-9999).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a block number out of the 0-9999 range has been programmed in the "RPT" or "GOTO" instruction.

SOLUTION Remember that the programming formats for these instructions are:
(RPT N(block number), N(block number))
(GOTO N(block number))
The block number (label) must be between 0 and 9999.

0063 'Program subroutine number 1 thru 9999.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a subroutine number out of the 0-9999 range has been programmed in the "SUB" instruction.

SOLUTION Remember that the programming format for this instruction is:
(SUB (integer))
The subroutine number must be between 0 and 9999.

0064 'Repeated subroutine.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE There has been an attempt to define a subroutine already existing in another program of the memory.

SOLUTION In the CNC memory, there could not be more than one subroutine with the same identifying number even if they are contained in different programs.

0065 'The main program cannot have a subroutine.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE/S The various causes might be:
1.- An attempt has been made to define a subroutine in the MDI execution mode.
2.- A subroutine has been defined in the main program.

SOLUTION The solution for each cause is:
1.- Subroutines cannot be defined from the «MDI execution» option of the menu.
2.- Subroutines must be defined after the main program or in a separate program. They cannot be defined before or inside the main program.

0066 'Expecting a message.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level, the "MSG" or "ERROR" instruction has been edited but without the message to be displayed.

SOLUTION Remember that the programming format of these instructions is:
(MSG "message")
(ERROR integer, "error message")
Although it can also be programmed like:
(ERROR integer)
(ERROR "error message")

0067 'OPEN is missing.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE While programming in high level, a "WRITE" instruction has been edited, but the OPEN instruction has not been written previously to tell it where that instruction has to be executed.

SOLUTION The "OPEN" instruction must be edited before the "WRITE" instruction to «tell» the CNC where (in which program) it must execute the "WRITE" instruction.

0069 'Program does not exist.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE In the «Pattern repeat canned cycle (G66)», «Roughing canned cycle along the X axis (G68)» or «Roughing canned cycle along the Z axis (G69)», the profiles have been programmed to be located in another program (parameter "Q"), but the program does not exist.

SOLUTION Parameter "Q" defines the program containing the definition of the cycle profiles. If this parameter is programmed, that program number must exist and it must contain the labels defined in parameters "S" and "E".

0070 'Program already exists.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE This error comes up during execution when using the "OPEN" instruction (While programming in high level language) to create an already existing program.

SOLUTION Change the program number or use parameters A/D in the "OPEN" instruction:

(OPEN P———,A/D,…)

Where:

- A: Appends new blocks after the existing ones.
- D: Deletes the existing program and it opens it as a new one.

0071 'Expecting a parameter'

DETECTED While editing tables.

CAUSE The wrong parameter number has been entered (possibly missing the "P" character) or an attempt has been made to carry out another action (move around in the table) before quitting the table editing mode.

SOLUTION Enter the number of the parameter to be edited or press [ESC] to quit this mode.

0072 'Parameter does not exist.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "ERROR" instruction has been edited, but the error number to be displayed has been defined either with a local parameter greater than 25 or with a global parameter greater than 299.

SOLUTION The parameters used by the CNC are:

- Local: 0-25
- -Global: 100-299

0075 'Read-only variable.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to assign a value to a read-only variable.

SOLUTION Read-only variables cannot be assigned any values through programming. However, their values can be assigned to a parameter.

0077 'Analog output not available.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to write to an analog output currently being used by the CNC.

SOLUTION The selected analog output may be currently used by an axis or a spindle. Select another analog output between 1 and 8.

0078 'Program channel 0(CNC),1(PLC) or 2(DNC).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "KEYSCR" instruction has been programmed, but the source of the keys is missing.

SOLUTION When programming the "KEYSCR" instruction, the parameter for the source of the keys must always be programmed:
(KEYSCR=0) : CNC keyboard
(KEYSCR=1) : PLC
(KEYSCR=2) : DNC
The CNC only lets modifying the contents of this variable if it is «zero»

0079 'Program error number 0 thru 9999.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "ERROR" instruction has been programmed, but the error number to be displayed is missing.

SOLUTION Remember that the programming format for this instruction is:
(ERROR integer, "error message")
Although it can also be programmed as follows:
(ERROR integer)
(ERROR "error message")

0081 'Incorrect expression.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, an expression has been edited with the wrong format.

SOLUTION Correct the block syntax.

0082 'Incorrect operation.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:
1.- While programming in high level language, the assignment of a value to a parameter is incomplete.
2.- While programming in high level language, the call to a subroutine is incomplete.

SOLUTION Correct (complete) the format to assign a value to a parameter or a call to a subroutine.

0083 'Incomplete operation.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S While programming in high level language, the "IF" instruction has been edited without the condition between brackets.

SOLUTION Remember that the programming format for this instruction are:
(IF (*condition*) <*action1*>)
(IF (*condition*) <*action1*> ELSE <*action2*>)
If the *condition* is true, it executes the <*action1*>, otherwise, it executes <*action2*>.

0084 'Expecting "=".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the "=" symbol in the right place.

0085 'Expecting ")".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the ")" symbol in the right place.

0086 'Expecting "(".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the "(" symbol in the right place.

0087 'Expecting ",".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.
2.- While programming in high level language, an ISO-coded instruction has been programmed.
3.- While programming in high level language, an operation has been assigned either to a local parameter greater than 25 or to a global parameter greater 299.

SOLUTION The solution for each cause is:

1.- Enter the "," symbol in the right place.
2.- A block cannot contain high level language instructions and ISO-coded instructions at the same time.
3.- The parameters used by the CNC are:
- Local: 0-25.
- Global: 100-299.

Other parameters out of this range cannot be used in operations.

0089 'Logarithm of zero or negative number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves the calculation of a negative number or a zero.

SOLUTION Only logarithms of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0090 'Square root of a negative number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves the calculation of the square root of a negative number.

SOLUTION Only the square root of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0091 'Division by zero.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves a division by zero.

SOLUTION Only divisions by numbers other than zero are allowed. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0092 'Base zero with positive exponent.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves elevating zero to a negative exponent (or zero).

SOLUTION Zero can only be elevated to positive exponents greater than zero. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0093 'Negative base with decimal exponent.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves elevating a negative number to a decimal exponent.

SOLUTION Negative numbers can only be elevated to integer exponents. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0094 'ASIN/ACOS range exceeded.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves calculating the arcsine or arccosine of a number out of the ±1 range.

SOLUTION Only the arcsine (ASIN) or arccosine (ACOS) of numbers between ±1 can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0095 'Program row number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, a window has been programmed with the "ODW" instruction, but the vertical position of the window on the screen is missing.

SOLUTION The vertical position of the window on the screen is defined by rows (0-25).

0096 'Program column number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, a window has been programmed with the "ODW" instruction, but the horizontal position of the window on the screen is missing.

SOLUTION The horizontal position of the window on the screen is defined by columns (0-79).

0097 'Program another softkey.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, the programming format for the "SK" instruction has not been respected.

SOLUTION Correct the syntax of the block. The programming format is:

(SK1=(text 1), SK2=(text 2)…)

If the "," character is entered after a text, the CNC expects the name of another softkey.

0098 'Program softkeys 1 thru 7.'

DETECTED While executing in the user channel.

CAUSE In the block syntax, a softkey has been programmed out of the 1 to 7 range.

SOLUTION Only softkeys within the 1 to 7 range can be programmed.

0099 'Program another window.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, the programming format for the "DW" instruction has not been respected.

SOLUTION Correct the syntax of the block. The programming format is:
(DW1=(assignment), DW2=(assignment)…)
If the "," character is entered after an assignment, the CNC expects the name of another window.

0100 'Program windows 0 thru 25.'

DETECTED While executing in the user channel.

CAUSE In the block syntax, a window has been programmed out of the 0 to 25 range.

SOLUTION Only windows within the 0 to 25 range can be programmed.

0101 'Program rows 0 thru 20.'

DETECTED While executing in the user channel.

CAUSE In the block syntax, a row has been programmed out of the 0 to 20 range.

SOLUTION Only rows within the 0 to 20 range can be programmed.

0102 'Program columns 0 thru 79.'

DETECTED While executing in the user channel.

CAUSE In the block syntax, a column has been programmed out of the 0 to 79 range.

SOLUTION Only columns within the 0 to 79 range can be programmed.

0103 'Program pages 0 thru 255.'

DETECTED While executing in the user channel.

CAUSE In the block syntax, a page has been programmed out of the 0 to 255 range.

SOLUTION Only pages within the 0 to 255 range can be programmed.

0104 'Program INPUT.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, an "IB" instruction has been edited without associating an "INPUT" to it.

SOLUTION Remember that the programming formats for this instruction are:
(IB (expression) = INPUT "text", format)
(IB (expression) = INPUT "text")

0105 'Program inputs 0 thru 25.'

DETECTED While executing in the user channel.

CAUSE In the block syntax, an input has been programmed out of the 0 to 25 range.

SOLUTION Only inputs within the 0 to 25 range can be programmed.

0106 'Program numerical format.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, an "IB" instruction has been edited with non-numeric format.

SOLUTION Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)
Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.
If the "," character is entered after the text, the CNC expects the format.

0107 'Do not program formats greater than 6.5 .'

DETECTED While executing in the user channel.

CAUSE While programming in high level language, an "IB" instruction has been edited in a format with more than 6 entire digits or more than 5 decimals.

SOLUTION Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)
Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

0108 'This command can only be executed in the user channel.'

DETECTED During execution.

CAUSE An attempt has been made to execute a block containing information that can only be executed through the user channel.

SOLUTION There are specific expressions for customizing programs that can only be executed inside the user program.

0109 'User channel: Do not program geometric aides, comp. or cycles'

DETECTED While executing in the user channel.

CAUSE An attempt has been made to execute a block containing geometric aide, tool radius/length compensation or machining canned cycles.

SOLUTION Inside a customizing program the following cannot be programmed:
- Neither geometric assistance nor movements.
- Neither tool radius nor length compensation.
- Canned cycles.

0110 'Local parameters not allowed.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Some functions can only be programmed with global parameters.

SOLUTION Global parameters are the ones included in the 100-299 range.

0111 'Block cannot be executed while running another program'

DETECTED While executing in MDI mode.

CAUSE An attempt has been made to execute a customizing instruction from MDI mode while the user channel program is running.

SOLUTION Customizing instructions can only be executed through the user channel.

0112 'WBUF can only be executed in user channel while editing'

DETECTED During execution or user channel execution.

CAUSE An attempt has been made to execute the "WBUF" instruction.

SOLUTION The "WBUF" instruction cannot be executed. It can only be used in the editing stage through the user input.

0113 'Table limits exceeded.'

DETECTED While editing tables.

CAUSE/S The various causes might be:
1.- In the tool offset table, an attempt has been made to define a tool offset with a greater number than allowed by the manufacturer.
2.- In the parameter tables, an attempt has been made to define a nonexistent parameter.

SOLUTION The tool offset number must be smaller than the one allowed by the manufacturer.

0114 'Offset: D3 X Z R F I K..'

DETECTED While editing tables.

CAUSE In the tool offset table, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order.

0115 'Tool: T4 D3 F3 N5 R5(.2).'

DETECTED While editing tables.

CAUSE In the tool table, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order.

0116 'Zero offset: G54-59 axes (1-5).'

DETECTED While editing tables.

CAUSE In the Zero offset table, the zero offset to be defined (G54-G59) has not be selected.

SOLUTION Enter the table parameters in the right order. To fill out the zero offset table, first select the offset to be defined (G54-G59) and then the zero offset position for each axis.

0117 'M function: M4 S4 bits(8).'

DETECTED While editing tables.

CAUSE In the «M» function table, the parameter editing order has not been respected.

SOLUTION Edit table following the format:
M1234 (associated subroutine) (customizing bits)

0118 'G51 [A] E'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE In the «Look-Ahead (G51)» function, the parameter for the maximum contouring error is missing.

SOLUTION This type of machining requires the programming of:
E : Maximum contouring error.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0119 'Leadscrew: Position-Error.'

DETECTED While editing tables.

CAUSE In the leadscrew compensation tables, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order

P123 (position of the axis to be compensated) (leadscrew error at that point)

0120 'Incorrect axis.'

DETECTED While editing tables.

CAUSE In the leadscrew compensation tables, an attempt has been made to edit a different axis from the one corresponding to that table.

SOLUTION Each axis has its own table for leadscrew compensation. The table for each axis can only contain the positions for that axis.

0121 'Program P3 = value.'

DETECTED While editing tables.

CAUSE In the machine parameter table, the editing format has not been respected.

SOLUTION Enter the table parameters in the right order.

P123 = (parameter value)

0122 'Magazine: P(1-255) = T(1-9999).'

DETECTED While editing tables.

CAUSE In the tool magazine table, the editing format has not been respected or some data is missing.

SOLUTION Enter the table parameters in the right order.

0123 'Tool T0 does not exist.'

DETECTED While editing tables.

CAUSE In the tool table, an attempt has been made to edit a tool as T0.

SOLUTION No tool can be edited as T0. The first tool must be T1.

0124 'Offset D0 does not exist.'

DETECTED While editing tables.

CAUSE In the tool table, an attempt has been made to edit a tool offset as D0.

SOLUTION No tool offset can be edited as D0. The first tool offset must be D1.

0125 'Do not modify the active tool or the next one.'

DETECTED During execution.

CAUSE In the tool magazine table, an attempt has been made to change the active tool or the next one.

SOLUTION During execution, neither the active tool nor the next one may be changed.

0126 'Tool not defined.'

DETECTED While editing tables.

CAUSE In the tool magazine table, an attempt has been made to assign to the magazine position a tool that is not defined in the tool table.

SOLUTION Define the tool in the tool table.

0127 'Magazine is not RANDOM.'

DETECTED While editing tables.

CAUSE There is no RANDOM magazine and, in the tool magazine table, the tool number does not match the tool magazine position.

SOLUTION When the tool magazine is not RANDOM, the tool number must be the same as the magazine position (pocket number).

0128 'The position of a special tool is set.'

DETECTED While editing tables.

CAUSE In the tool magazine table, an attempt has been made to place a tool in a magazine position reserved for a special tool.

SOLUTION When a special tool occupies more than one position in the magazine, it has a reserved position in the magazine. No other tool can be placed in this position.

0129 'Next tool only possible in machining centers.'

DETECTED During execution.

CAUSE A tool change has been programmed with M06 and the machine is not a machining center (it is not expecting the next tool).

SOLUTION When the machining is not a machining center, the tool change is done automatically when programming the tool number «T».

0130 'Write 0/1.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values of 0 or 1.

0131 'Write +/-.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values of + or -.

0132 'Write YES/NO.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values of YES or NO.

0133 'Write ON/OFF.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values of ON or OFF.

0134 'Values 0 thru 2.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values between 0 and 2.

0135 'Values 0 thru 3.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 3.

0136 'Values 0 thru 4.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 4.

0137 'Values 0 thru 9.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 9.

0139 'Values 0 thru 100.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 100.

0140 'Values 0 thru 255.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 255.

0141 'Values 0 thru 9999.'

DETECTED While editing machine parameters
CAUSE/S The various causes might be:
1.- An attempt has been made to assign the wrong value to a parameter.
2.- During execution, when inside the program a call has been to a subroutine (MCALL, PCALL) greater than 9999.
SOLUTION The solution for each cause is:
1.- The parameter only admits values between 0 and 9999.
2.- The subroutine number must be between 1 and 9999.

0142 'Values 0 thru 32767.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 32767.

0144 'Values 0 thru 65535.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 65535.

0145 'Format +/- 5.5.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values with the format: ± 5.5.

0147 'Numerical format exceeded.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A data or parameter has been assigned a value greater than the established format.

SOLUTION Correct the syntax of the block. Most of the time, the numeric format will be 5.4 (5 integers and 4 decimals).

0148 'Text too long.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "ERROR" or "MSG" instruction has been assigned a text with more than 59 characters.

SOLUTION Correct the syntax of the block. The "ERROR" and "MSG" instructions cannot be assigned texts longer than 59 characters.

0149 'Incorrect message.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the text associated with the "ERROR" or "MSG" instruction has been edited wrong.

SOLUTION Correct the syntax of the block. The programming format is:
(MSG "message")
(ERROR number, "message")
The message must be between " ".

0150 'Incorrect number of bits.'

DETECTED While editing tables.

CAUSE/S The various causes might be:
1.- In the «M» function table, in the section on customizing bits:
- The number does not have 8 bits.
- The number does not consist of 0's and 1's.
2.- In the machine parameter table, an attempt has been made to assign the wrong value of bit to a parameter.

SOLUTION The solution for each cause is:
1.- The customizing bits must consist of 8 digits of 0's and 1's.
2.- The parameter only admits 8-bit or 16-bit numbers.

0152 'Incorrect parametric programming.'

DETECTED During execution.

CAUSE The parameter has a value that is incompatible with the function it has been assigned to.

SOLUTION This parameter may have taken the wrong value, in the program history. Correct the program so this parameter does not reach the function with that value.

0154 'Insufficient memory.'

DETECTED During execution.

CAUSE The CNC does not have enough memory to internally calculate the paths.

SOLUTION Sometimes, this error is taken care of by changing the machining conditions.

0156 'Don't program G33 ,G95 or M19 S with no spindle encoder'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A "G33", "G95" or "M19 S" has been programmed without having an encoder on the spindle.

SOLUTION If the spindle does not have an encoder, functions "M19 S", "G33" or "G95". Spindle machine parameter "NPULSES (P13)" indicates the number of encoder pulses per turn.

0159 'Inch programming limit exceeded.'

DETECTED During execution.

CAUSE An attempt has been made to execute in inches a program edited in millimeters.

SOLUTION Enter function G70 (inch programming) or G71 (mm programming) at the beginning of the program.

0162 'No negative radius allowed with absolute coordinates'

DETECTED During execution.

CAUSE While operating with absolute polar coordinates, a movement with a negative radius has been programmed.

SOLUTION Negative radius cannot be programmed when using absolute polar coordinates.

0164 'Wrong password.'

DETECTED While assigning protections.

CAUSE [ENTER] has been pressed before selecting the type of code to be assigned a password.

SOLUTION Use the softkeys to select the type of code to which a password is to be assigned.

0165 'Password: use uppercase/lowercase letters or digits.'

DETECTED While assigning protections.

CAUSE A bad character has been entered in the password.

SOLUTION The password can only consist of letters (upper and lower case) or digits.

0166 'Only one HIRTH axis per block is allowed.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A movement has been programmed which involves the movement of two HIRTH axes simultaneously.

SOLUTION Only one HIRTH axis can be moved at a time.

0167 'Position-only rotary axis: Absolute values 0 - 359.9999'

DETECTED During execution.

CAUSE A movement of a positioning-only rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

SOLUTION Positioning-only rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0168 'Rotary axis: Absolute values (G90) within +/-359.9999.'

DETECTED During execution.

CAUSE A movement of a rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

SOLUTION Rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0169 'Modal subroutines cannot be programmed.'

DETECTED While executing in MDI mode

CAUSE An attempt has been made to call upon a modal subroutine (MCALL).

SOLUTION MCALL modal subroutines cannot be executed from the menu option «MDI execution».

0171 'The window must be previously defined.'

DETECTED During normal execution or execution through the user channel.

CAUSE An attempt has been made to write in a window (DW) that has not been previously defined (ODW).

SOLUTION It is not possible to write in a window that has not been previously defined. Check that the window to write in (DW) has been previously defined.

0172 'The program is not accessible'

DETECTED During execution.

CAUSE An attempt has been made to execute a program that cannot be executed.

SOLUTION The program may be protected against execution. To know if the program can be executed, check the attributes column, if the letter «X» is missing, it means that it cannot be executed.

0174 'Circular (helical) interpolation not possible.'

DETECTED During execution.

CAUSE An attempt has been made to execute a helical interpolation while the «LOOK-AHEAD (G51)» function was active.

SOLUTION Helical interpolations are not possible while the «LOOK-AHEAD (G51)» function is active.

0175 'Analog inputs: ANAI(1-8) = +/-5 Volts.'

DETECTED During execution.

CAUSE An analog input has taken a value out of the ±5V range.

SOLUTION Analog inputs may only take values within the ±5V range.

0176 'Analog outputs: ANAO(1-8) = +/-10 Volts.'

DETECTED During execution.

CAUSE An analog output has been assigned a value out of the ±10V range.

SOLUTION Analog outputs may only take values within the ±10V range.

0178 'G96 only possible with analog spindle.'

DETECTED During execution.

CAUSE The "G96" function has been programmed but either the spindle speed is not controlled or the spindle does not have an encoder.

SOLUTION To operate with the "G96" function, the spindle speed must be controlled (SPDLTYPE(P0)=0) and the spindle must have an encoder (NPULSES(P13) other than zero).

0180 'Program DNC1/2, HD or CARD A (optional).'

DETECTED While editing or executing.

CAUSE While programming in high level language, in the "OPEN" and "EXEC" instructions, an attempt has been made to program a parameter other than DNC1/2, HD or CARD A, or the DNC parameter has been assigned a value other than 1 or 2.

SOLUTION Check the syntax of the block.

0181 'Program A (append) or D (delete).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "OPEN" instruction the A/D parameter is missing.

SOLUTION Check the syntax of the block. The programming format is:
(OPEN P———,A/D,…)
Where:
- A : Appends new blocks after the existing ones.
- D : Deletes the existing program and it opens it as a new one.

0182 'Option not available.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A «G» function has been defined which is not a software option.

0185 'Tool offset does not exist'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Within the block syntax, a tool offset has been called upon which is greater than the ones allowed by the manufacturer.

SOLUTION Program a new smaller tool offset.

0186 'There is no C axis.'

DETECTED While editing or executing programs via DNC.

CAUSE An attempt has been made to activate the «C» axis, but the machine does not have a «C» axis.

0187 'G66,G68,G69 not allowed when machining with C axis.'

DETECTED While executing.

CAUSE An attempt has been made to execute a canned cycle "G66", "G67" or "G68" while the «C» axis is active.

SOLUTION To operate with these cycle, deactivate the «C» axis.

0188 'Function not possible from PLC.'

DETECTED During execution.

CAUSE From the PLC channel and using the "CNCEX" instruction, an attempt has been made to execute a function that is incompatible with the PLC channel execution.

SOLUTION The installation manual (chapter 11.1.2) offers a list of the functions and instructions that may be executed through the PLC channel.

0189 'There is no live tool.'

DETECTED While editing or executing programs via DNC.

CAUSE An attempt has been made to start the live tool "M45 S—" but the machine does not have a live tool.

0194 'Repositioning not allowed.'

DETECTED During execution.

CAUSE The axes cannot be repositioned using the "REPOS" instruction because the subroutine has not been activated with one of the interruption inputs.

SOLUTION Before executing the "REPOS" instruction, one of the interruption inputs must be activated.

0195 'Axes X,Y or Z slaved or synchronized.'

DETECTED During execution.

CAUSE While programming in high level language, an attempt has been made to execute a probing cycle using the "PROBE" instruction, but one of the X or Z axis is slaved or synchronized.

SOLUTION To execute the "PROBE"¨ instruction, the X-Z axes must not be slaved or synchronized. To unslave the axes, program "G78".

0196 'Axes X,Y and Z must exist.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, an attempt has been made to edit the "PROBE" instruction, but one of the X or Z axis is missing.

SOLUTION To operate with the "PROBE" instruction, the X-Z axes must be defined.

0197 'Program G15 before C axis.'

DETECTED While editing or executing programs via DNC.

CAUSE An attempt has been made to execute an operation on the «C» axis, but it was not active.

SOLUTION To work with the «C» axis, it must be activated first with function "G15".

0199 'Preset of rotary axes: Values between 0-359.9999. '

DETECTED While presetting coordinates.

CAUSE An attempt has been made preset the coordinates of a rotary axis with a value out of the 0 to 359.9999 range.

SOLUTION The preset value of rotary axes must be within the 0 to 359.9999 range.

0200 'Program: G52 axis +/-5.5.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE When programming the «Movement against a hard stop (G52)», either the axis to be moved has not been programmed or several axes have been programmed.

SOLUTION When programming the "G52" function, the axis to be moved must be programmed but only one axis at a time.

0206 'Values 0 thru 6.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values between 0 and 6.

0207 'Complete Table.'

DETECTED While editing tables.

CAUSE In the tables for «M» functions or tool offsets, an attempt has been made to define more data than those allowed by the manufacturer by means of machine parameters. When loading a table via DNC, the CNC does not delete the previous table, it replaces the existing values and it copies the new data in the free positions of the table.

SOLUTION The maximum number of data that can be defined is limited by the machine parameters:

- Maximum number of «M» functions : NMISCFUN(P29).
- Maximum number of : NTOOL(P23).
- Maximum number of tool offset : NTOFFSET(P27).
- Maximum number of magazine positions : NPOCKET(P25).

To load a new table via DNC, the previous table should be deleted.

0208 'Program A from 0 to 255'

DETECTED During execution.

CAUSE In the «LOOK-AHEAD (G51)» function, parameter "A" (% of acceleration to be applied) has been programmed with a value greater than 255.

SOLUTION Parameter "A" is optional, but when programmed, it must have a value between 0 and 255.

0209 'Program nesting not allowed.'

DETECTED During execution.

CAUSE From a running program, an attempt has been made to execute another program with the "EXEC" instruction which in turn also has an "EXEC" instruction.

SOLUTION Another program cannot be called upon from a program being executed using the "EXEC" instruction.

0210 'No compensation is permitted.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to activate or cancel tool radius compensation (G41, G42, G40) in a block containing a nonlinear movement.

SOLUTION Tool radius compensation must be activated/deactivated in linear movements (G00, G01).

0213 'For G28 or G29, a second spindle is required.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to select the work spindle with "G28/G29", but the machine only has one work spindle.

SOLUTION If the machine only has one work spindle, the "G28/ G29" functions cannot be programmed.

0214 'Invalid G function when selecting a profile'

DETECTED While restoring a profile.

CAUSE Within the group of blocks selected to restore the profile, there is a block containing a «G» code that does not belong in the profile definition.

SOLUTION The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08
G09	G36	G37	G38	G39	G90
G91	G93				

0215 'Invalid G function after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a block containing a «G» function that does not belong in the profile definition.

SOLUTION The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08
G09	G36	G37	G38	G39	G90
G91	G93				

0216 'Nonparametric assignment after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, a nonparametric assignment has been programmed in high level language (a local or global parameter).

SOLUTION The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0217 'Invalid programming after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a high level block that is not an assignment.

SOLUTION The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0218 'The axis cannot be programmed after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, a position has been defined on an axis that does not belong to the active plane. A surface coordinate may have been defined after the starting point of the profile.

SOLUTION The surface coordinate of the profiles is only defined in the starting block of the first profile, the one corresponding to the starting point of the outside profile.

0219 'First point programmed wrong when selecting profile'

DETECTED While selecting a profile.

CAUSE The starting point of the profile has been programmed wrong. One of the two coordinates defining its position is missing.

SOLUTION The starting point of a profile must be defined on the two axes forming the active plane.

0227 'Program Q between +/-359.9999.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Electronic threading (G33)» function, the entry angle "Q" has been programmed with a value out of the ±359.9999 range.

SOLUTION Program an entry angle within the ±359.9999 range.

0228 'Do not program "Q" with parameter M19TYPE=0.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Electronic threading (G33)» function, an entry angle "Q" has been programmed, but the type of spindle orientation available does not allow this operation.

SOLUTION In order to define an entry angle, spindle machine parameter M19TYPE(P43) must be set to «1».

0229 'Program maximum Z'
0230 'Program inside R'
0231 'Program outside R'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, in the "DGWZ" instruction, the indicated limit is missing or it has been defined with a non-numerical value.

SOLUTION Check the syntax of the block.

0234 'Wrong graphic limits'

DETECTED During execution.

CAUSE One of the lower limits defined with the "DGWZ" instruction is greater than its corresponding upper limit.

SOLUTION Program the upper limit of the graphics display area greater than the lower ones.

PREPARATION AND EXECUTION ERRORS

1000 'Not enough information about the path'

DETECTED During execution.

CAUSE The program has too many consecutive blocks without path data to apply tool radius compensation, rounding, chamfers or tangential entry / exit.

SOLUTION In order to carry out these operations, the CNC needs to know the path in advance; therefore, there cannot be more than 48 consecutive blocks without the path to be followed.

1001 'Plane change during rounding or chamfering'

DETECTED During execution.

CAUSE A plane change has been programmed on the path following a «Controlled corner rounding (G36)» or a «Chamfer (G39)».

SOLUTION The plane cannot be changed while executing a rounding or a chamfer. The path following the definition of a corner rounding or chamfer must be in the same plane as the rounding or chamfer.

1002 'Rounding radius too large '

DETECTED During execution.

CAUSE In the «Controlled corner rounding (G36)» function, a rounding radius has been programmed larger than one of the paths where it is defined.

SOLUTION The rounding radius must be smaller than the paths defining it.

1003 'Rounding in last block'

DETECTED During execution.

CAUSE A «Controlled corner rounding (G36)» or a «Chamfer (G39)» has been defined on the last path of the program or when the CNC cannot find information about the path following the definition of the corner rounding or chamfer.

SOLUTION A corner rounding or chamfer must be defined between two paths.

1004 'Tangential exit programmed incorrectly'

DETECTED During execution.

CAUSE The movement following a tangential exit (G38) is a circular path.

SOLUTION The movement following a tangential exit (G38) must be straight line.

1005 'Chamfer programmed incorrectly'

DETECTED During execution.

CAUSE The movement following a chamfer (G39) is a circular path.

SOLUTION The movement following a chamfer (G39) must be a straight line.

1006 'Chamfer value too large'

DETECTED During execution.

CAUSE In the «Chamfer (G39)» function, a chamfer has been programmed larger than the paths where it has been defined.

SOLUTION The chamfer must be smaller than the paths defining it.

1007 'G8 defined incorrectly'

DETECTED During execution.

CAUSE/S The probable causes might be:
1.- When programming a full circle with the «Arc tangent to previous path (G08)» function.
2.- When the tangent path ends at one point of the previous path or on its extension (In straight line).
3.- While operating with an irregular pocket with islands, when programming a "G08" function in the block following the definition of the beginning of the profile (G00).

SOLUTION The solution for each cause is:
1.- Full circles cannot be programmed using function "G08".
2.- The tangent path cannot end at one point of the previous path or on its extension (In straight line).
3.- The CNC does not have information about the previous path and it cannot execute a tangent arc.

1008 'There is no information on previous path'

DETECTED During execution.

CAUSE An arc tangent to the previous path has been programmed with "G08", but there isn't enough information about the previous path.

SOLUTION In order to make a path tangent to a previous one, there must be information about the previous path and it must be in the 48 blocks prior to the tangent path.

1010 'Wrong plane in tangential path'

DETECTED During execution.

CAUSE A plane change has been programmed between the definition of the «Arc tangent to previous path (G08)» function and the previous path.

SOLUTION The plane change cannot be done between both paths.

1011 'The radius has not been programmed for G15.'

DETECTED While executing

CAUSE The Z-C plane has been selected, but the radius of the cylinder to be machined has not been defined.

SOLUTION To work in the Z-C plane, first, the radius of the cylinder to be machined must be defined with function "G15 R—"

1015 'Tool not defined in tool table'

DETECTED During execution.

CAUSE A tool change has been defined, but the new tool is not defined in the tool table.

SOLUTION Define the new tool in the tool table.

1016 'The tool is not in the tool magazine'

DETECTED During execution.

CAUSE A tool change has been defined, but the new tool is not defined in any table position of the tool magazine.

SOLUTION Define the new tool in the tool magazine table.

1017 'There is no empty pocket in the tool magazine'

DETECTED During execution.

CAUSE A tool change has been defined, but there isn't any pockets in the magazine to place the tool that currently is in the spindle.

SOLUTION The new tool may be defined in the tool table as special and more than magazine position may be reserved for it. In that case, that position is fixed for that tool and it cannot be occupied by another tool. To avoid this error message, a free position should be left in the tool magazine.

1018 'A tool change has been programmed without M06'

DETECTED During execution.

CAUSE After searching for a tool and before searching for the next one, an M06 has not been programmed.

SOLUTION This error comes up when having a machining center (general machine parameter TOFFM06(P28)=YES) which has a cyclic automatic tool changer (general machine parameter CYCATC(P61)=YES). In that case, after searching for a tool and before searching for the next one, a tool change has to be made using an M06.

1019 'There is no tool of the same family to replace it'

DETECTED During execution.

CAUSE The real life of the requested tool exceeds its nominal life. The CNC has tried to replace it with another one of the same family (type), it has found none.

SOLUTION Replace the tool or define another one of the same family.

1020 'Do not use high level to change active tool or next one'

DETECTED During execution.

CAUSE While programming in high level language using the "TMZT" variable, an attempt has been made to assign the active tool (or the next one) to a magazine position.

SOLUTION To change the active tool or the next one, use the «T» function. The active tool or the next one cannot be moved to the magazine using the "TMZT" variable.

1021 'The canned cycle is missing a tool offset'

DETECTED During execution.

CAUSE A probing canned cycle "PROBE" has been programmed for tool calibration, but no tool offset has been selected.

SOLUTION To execute the «Tool calibration canned cycle (PROBE)», the tool offset that is supposed to store the data of the probing cycle must be previously selected.

1028 'Do not switch axes over or back while G15, G23, G48 or G49 are active'

DETECTED During execution.

CAUSE An attempt has been made to switch an axis or switch it back (G28/G29) while the "G15", "G23", "G48" or "G49" function was active.

SOLUTION The axes cannot be switched while the "G15", "G23", "G48", "G49" are active.

1029 'Do not switch axes already switched over'

DETECTED During execution.

CAUSE An attempt has been made to switch an axis (G28) which is already switched with another one.

SOLUTION An axis switched with another one cannot be directly switched with a third one. It must be switched back first. (G29 axis).

1030 'Not enough room for the automatic range change M code'

DETECTED During execution.

CAUSE While using an automatic gear change and having programmed in a block seven «M» functions and an «S» function involving a tool change, the CNC cannot include the «M» for the automatic tool change in that block.

SOLUTION Program one of the «M» functions or the «S» function in a separate block.

1031 'A subroutine is not allowed for automatic range change'

DETECTED During execution.

CAUSE In machines using an automatic gear change, when programming an «S» speed that involves a gear change and the «M» function for the automatic gear change has a subroutine associated with it.

SOLUTION When using an automatic gear change, the «M» functions for the gear change cannot have an associated subroutine.

1032 'Spindle speed range not defined for M19'

DETECTED During execution.

CAUSE An "M19" has been programmed, but none of the gear change functions is active ("M41", "M42", "M43" or "M44").

SOLUTION On power-up, the CNC does not assume any gear. Therefore, if the gear change function is not automatically generated (spindle parameter AUTOGEAR(P6)=NO), the auxiliary functions must be programmed for the gear change ("M41", "M42", "M43" or "M44").

1033 'Incorrect range change'

DETECTED During execution.

CAUSE/S The various probable causes are:

1.- When trying to make a gear change and the machine parameters for the gears (MAXGEAR1, MAXGEAR2, MAXGEAR3, or MAXGEAR4) are set wrong. All the gears have not be used and the unused ones have been set to maximum speed of zero.

2.- When a gear change has been programmed ("M41", "M42", "M43" or "M44"), but the PLC has not responded with corresponding active gear signal (GEAR1, GEAR2, GEAR3 or GEAR4).

SOLUTION The solution for each cause is:

1.- When not using all four gears, the lowest ones must be used starting with "MAXGEAR1", and the unused gears must be assigned the highest value of the ones used.

2.- Check the PLC program.

1034 'S has been programmed without an active range'

DETECTED During execution.

CAUSE An attempt has been made to start the spindle, but no gear has been selected.

SOLUTION On power-up, the CNC does not assume any gear. Therefore, if the gear change function is not automatically generated (spindle parameter AUTOGEAR(P6)=NO), the auxiliary functions must be programmed for the gear change ("M41", "M42", "M43" or "M44").

1035 'S programmed too large'

DETECTED During execution.

CAUSE An «S» value has been programmed that is greater than the maximum value allowed for the last active range (gear).

SOLUTION Program a smaller «S» value.

1036 'S not programmed in G95 or threadcutting'

DETECTED During execution.

CAUSE Either the feedrate has been programmed in mm (inches) per rev. (G95) or the «Electronic threading (G33)» without having a spindle speed selected.

SOLUTION Working in mm/rev. (G95) or making an thread (using G33) requires the programming of an "S" speed.

1037 'S has not been programmed in G96'

DETECTED While executing.

CAUSE The «Constant Surface Speed (G96)» function has been programmed, but a cutting speed has not been programmed or a previous one exists or no spindle gear (range) has been selected.

SOLUTION To work at constant surface speed (G96), a cutting speed "S" must be programmed and a spindle range (gear) must be active.

1040 'Canned cycle does not exist'

DETECTED During execution in MDI mode.

CAUSE An attempt has been made to execute a canned cycle (G8x) after interrupting a program while executing a canned cycle (G8x) and then doing a plane change.

SOLUTION Do not interrupt the program while executing a canned cycle.

1042 'Invalid parameter value in canned cycle'

DETECTED While executing.

CAUSE When defining a canned cycle, a parameter has been defined with the wrong value. Maybe, a negative or zero value has been assigned to a parameter that only admits positive values.

SOLUTION Correct the parameter definition:

- In the «Pattern repeat canned cycle»:
  - Parameter "C" only admits positive values greater than zero.
  - Parameter "A" only admits a value of "0" or "1".
  - Parameter "J" only admits positive values greater than zero.
- In the «Roughing canned cycle along the Z axis» or «Roughing canned cycle along the X axis», parameter "C" only admits positive values greater than zero.
- In the «Axial drilling / tapping canned cycle»:
  - Parameter "I" only admits values other than zero.
  - Parameter "B" only admits positive values or zero.
- In the «Facing canned cycle with arcs» or «Turning canned cycle with arcs», parameter "C" only admits positive values greater than zero.
- In the «Face threading canned cycle» or «Longitudinal threading canned cycle», parameter "I", "B", "E" or "C" has been defined with zero value.
- In the «Grooving canned cycle along the Z axis» or «Grooving canned cycle along the X axis», parameter "C" only admits positive values greater than zero.
- In the «radial drilling / tapping » or «axial drilling / tapping» canned cycles:
  - Parameter "I" only admits values other than zero.
  - Parameter "B" only admits positive values or zero.
  - Parameter "J" only admits positive values greater than zero.
- In the «radial slot milling» or «Axial slot milling» canned cycles, the dimension of the slot cannot be zero and parameters "I" and "J" only take positive values greater than zero.

1043 'Invalid tool for programmed profile.'

DETECTED While executing.

CAUSE The selected tool cannot machine any part of the profile.

SOLUTION Choose another more appropriate tool to machine the profile.

1044 'A self-intersecting contour has been programmed.'

DETECTED While executing.

CAUSE Within a set of profiles, there one that intersects itself.

SOLUTION Check the definition of the profiles. A profile cannot intersect itself.

1045 'Invalid cutter geometry angle'

DETECTED While executing

CAUSE An invalid value has been assigned to the angles of the cutter geometry.

SOLUTION Correct the tool geometry data.

1046 'Wrong tool position prior to canned cycle'

DETECTED While executing.

CAUSE The canned cycle calling point has been defined wrong.

SOLUTION The canned cycle calling point must be located outside the part at a distance greater than the one defined as the finishing stock on both axes. (In cycles not having finishing stock, the safety distance must be used).

1047 'Location code not allowed in canned cycle'

DETECTED While executing.

CAUSE The location code (tool shape) is not the right one to execute the machining operation.

SOLUTION Choose a tool with the right location code to carry out the machining operation.

1048 'Invalid cutter width'

DETECTED While executing.

CAUSE A grooving operation has been defined with a cutter width value of zero.

SOLUTION Check the definition of the cutter dimensions (NOSEW). The cutter width must be other than zero.

1049 'Incompatible tool position and tool code in profile cycle'

DETECTED While executing.

CAUSE The canned cycle calling point is defined wrong or the tool shape or location code is not the right one to carry out the machining operation.

SOLUTION The canned cycle calling point must be located outside the part at a distance greater than the one defined as the finishing stock on both axes. On the other hand, the tool location code must allow making the profile without running into the part.

1050 'Incorrect variable value'

DETECTED During execution.

CAUSE Too high a value has been assigned to a variable by means of parameters.

SOLUTION Check the program history, and make sure that that parameter does not reach the assignment block with that value.

1051 'Incorrect access to PLC variables'

DETECTED During execution.

CAUSE An attempt has been made to read a PLC variable from the CNC, but it was not defined in the PLC program.

1052 'Access to a variable with non-permitted index'

DETECTED While editing.

CAUSE While programming in high level language, an operation is carried out with either a local parameter greater than 25 or with a global parameter greater than 299.

SOLUTION The CNC uses the following parameters:
- Local: 0-25.
- Global: 100-299.

No other parameters can be used in the operations.

1053 'Local parameters not accessible'

DETECTED During execution in the user channel.

CAUSE An attempt has been made to execute a block containing an operation with local parameters.

SOLUTION The program executed in the user channel cannot carry out operations with local parameters (P0 through P25).

1054 'Local parameters not accessible'

DETECTED During execution.

CAUSE While programming in high level language, more than 6 nesting levels have been used with the "PCALL" statement within the same loop.

SOLUTION No more than 6 nesting levels are possible with local parameters within the 15 nesting levels for subroutines. Every time a call is made with the "PCALL" statement, a new nesting loop is generated for local parameters as well as for the subroutines.

1055 'Nesting exceeded.'

DETECTED During execution.

CAUSE While programming in high level language, more than 15 nesting levels have been used with the "CALL", "PCALL" or "MCALL" statements within the same loop.

SOLUTION No more than 15 nesting levels are possible. Every time a called is made with the "CALL", "PCALL" or "MCALL" statements, a new nesting level is generated.

1056 'RET not associated to a subroutine'

DETECTED During execution.

CAUSE The "RET" instruction has been edited without having previously edited the "SUB" instruction.

SOLUTION To use the "RET" instruction (end of subroutine), the subroutine must start with the "SUB" instruction (subroutine number).

1057 'Subroutine not defined'

DETECTED During execution.

CAUSE A call has been made (CALL, PCALL…) to a subroutine that is not defined in the CNC's memory.

SOLUTION Check that the name of the subroutine is correct and that it exists in the CNC's memory (not necessarily in the same program making the call).

1058 'Probing canned cycle not defined'

DETECTED While executing.

CAUSE An unavailable probing canned cycle has been defined with the "PROBE" instruction

SOLUTION The probing canned cycles available with the "PROBE" instruction are 1 through 4.

1059 'Jump to an undefined label'

DETECTED During execution.

CAUSE While programming in high level language, the "GOTO N—" instruction has been programmed, but the programmed block number (N) does not exist.

SOLUTION When programming the "GOTO N—" instruction, the block it refers to must be defined in the same program.

1060 'Label not defined'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- While programming in high level language, the "RPTN—, N—" instruction has been programmed, but the programmed block number (N) does not exist.

2.- While in the «Pattern repeat canned cycle (G66)», «Roughing canned cycle along the X axis (G68)» or «Roughing canned cycle along the Z axis (G69)» a profile has been programmed, but one of the data defining the beginning (S) or end (E) of the profiles is missing.

SOLUTION The solution for each cause is:

1.- When programming the "RPTN—, N—" instruction, the block it refers to must be defined in the same program.

2.- Check the program. Edit the label for the "S" parameter at the beginning of the profile definition and the label for the "E" parameter at the end of the profile definition.

1061 'Label cannot be searched'

DETECTED During execution in MDI mode

CAUSE While programming in high level language, an "RPT N—, N—" or "GOTO N—" instruction has been defined

SOLUTION "RPT" or "GOTO" type instructions cannot be programmed in MDI mode.

1062 'Subroutine not available in program'

DETECTED During execution.

CAUSE A subroutine has been called which is contained in a program that is currently being used by the DNC.

SOLUTION Wait for the DNC to be done with the program, If the subroutine is going to be used often, it is advisable to keep it in a separate program.

1063 'Program cannot be opened.'

DETECTED During execution.

CAUSE While running a program in infinite mode, an attempt has been made to execute another infinite program using the "EXEC" instruction at the running program.

SOLUTION Only one infinite program may be run at a time.

1064 'The program cannot be executed.'

DETECTED During execution.

CAUSE An attempt has been made to execute a program from another one using the "EXEC" instruction, but the program does not exit or is protected against execution.

SOLUTION The program to be executed with the "EXEC" instruction must be in CNC memory and it must be executable.

1065 'Beginning of compensation without a straight path'

DETECTED During execution.

CAUSE The first movement in the work plane after activating tool radius compensation (G41/G42) is not a linear movement.

SOLUTION The first movement after activating tool radius compensation (G41/G42) must be a linear movement.

1066 'End of compensation without a straight path'

DETECTED During execution.

CAUSE The first movement in the work plane after canceling tool radius compensation (G40) is not a linear movement.

SOLUTION The first movement after canceling tool radius compensation (G40) must be a linear movement.

1067 'Compensation radius too large'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42) an inside arc has been programmed with a radius smaller than the tool radius.

SOLUTION Use a tool with a smaller radius. When working with tool radius compensation (G41/G42), the arc radius must be greater than the tool radius. Otherwise, the tool cannot machine along the programmed path

1068 'Step in a straight path'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42), the profile has a straight section that cannot be machined because the tool diameter is too large.

SOLUTION Use a tool with a smaller radius.

1070 'Step in circular path'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42), the profile has a circular section that cannot be machined because the tool diameter is too large.

SOLUTION Use a tool with a smaller radius.

1071 'Compensation plane change'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42), another work plane has been selected.

SOLUTION To change the work plane, tool radius compensation must be canceled (G40).

1072 'Radius comp. not possible when positioning rotary axis'

DETECTED During execution.

CAUSE An attempt has been made to move a positioning-only rotary axis while tool radius compensation (G41/G42) is on.

SOLUTION Positioning-only rotary axes do not admit tool radius compensation. To cancel it, use the "G40" function.

1076 'Angle coordinate programmed incorrectly'

DETECTED During execution.

CAUSE While programming in the «angle-coordinate» format, an axis movement has been programmed with an angle perpendicular to that axis (v.g. the main plane is formed by the X, Z axes and the X axis is programmed to move at 90º).

SOLUTION Check and correct the definition of the movement in the program. When working with parameters, check that they reach the definition of the movement with the right values.

1077 'Arc programmed with radius too small or complete circle'

DETECTED During execution.

CAUSE/S The probable causes might be:
1.- When programming a full circle with the format: "G02/G03 X Z R".
2.- When programming with the format "G02/G03 X Z R", but the distance to the arc's end point is greater than the diameter of the programmed circle.

SOLUTION The solution for each cause is:
1.- With this format, full circles cannot be made. Program the end point with different coordinates from those of the starting point.
2.- The diameter of the circle must be greater than the distance to the arc's end point.

1078 'Negative radius in polar coordinates'

DETECTED During execution.

CAUSE While working in incremental polar coordinates, a block is executed which gives a negative final radius position.

SOLUTION When programming incremental polar coordinates, negative radius can be programmed, but the final (absolute) position of the radius must be positive.

1079 'There is no subroutine associated with G74'

DETECTED While executing a home search.

CAUSE/S The probable causes might be
1.- When trying to carry out a home search (on all axes) manually, but the associated subroutine indicating the searching sequence does not exist.
2.- Function "G74" has been programmed, but the associated subroutine indicating the searching sequence does not exist.

SOLUTION The solution for each cause is:
1.- To execute function "G74", its associated subroutine must be defined.
2.- If function "G74" is to be executed from a program, the home searching sequence for the axes may be defined.

1080 'Plane change during tool inspection'

DETECTED While executing the «tool inspection» option.

CAUSE The work plane has been changed, but it has not been restored before resuming execution.

SOLUTION Before resuming execution, the plane that was active before doing the «tool inspection» must be restored.

1081 'Block not allowed in MDI or during tool inspection'

DETECTED While executing the «tool inspection» option.

CAUSE An attempt has been made to execute the "RET" instruction.

SOLUTION This instruction cannot be executed within the «tool inspection» option.

1082 'Probe signal has not been received'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- A "PROBE" probing canned cycle has been programmed, but the probe has moved the maximum safety distance of the cycle without sending the probe signal to the CNC.

2.- When programming the "G75" function, the end coordinate has been reached without receiving the probe signal. (Only when general machine parameter PROBERR(P119)=YES).

SOLUTION The solution for each cause is:

1.- Check that the probe is connected properly.
The maximum probing distance (in the PROBE cycles) depends on the safety distance "B". To increase this distance, increase the safety distance.

2.- If PROBERR(P119)=NO, no error will be issued when this end coordinate is reached without receiving the probe signal (only with the "G75" function).

1083 'Range exceeded'

DETECTED During execution.

CAUSE The distance to travel for the axes very long and the programmed feedrate for that movement is very low.

SOLUTION Program a higher feedrate for this movement.

1084 'Circular path programmed incorrectly'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- When programming an arc using the format: "G02/G03 X Y I J", an arc cannot be made with the programmed radius and end point.

2.- When programming an arc using the format: "G09 X Y I J", The three points of the arc are in line or there are identical points.

3.- When trying to make a rounding or a tangential entry on a path not belonging to the active plane.

4.- When programming a tangential exit and the next path is tangent to (and on the linear extension of) the one prior to the tangential exit.

If the error comes up in the block calling the «Pattern repeat canned cycle (G66)», «Roughing canned cycle along the X axis (G68)» or «Roughing canned cycle along the Z axis (G69)», it is because one of the cases mentioned earlier occur in the set of blocks defining the profiles.

SOLUTION The solution for each cause is:

1.- Correct the syntax of the block. The coordinates of the end point or of the radius are defined wrong.

2.- The three points used to define the arc must be different and cannot be in line.

3.- Maybe a plane has been defined using "G16", "G17", "G18" or "G19". In that case, rounding, chamfers, and tangential entries/exits can be carried out on the main axes defining that plane. To make them in another plane, it must be selected before.

4.- The path after the tangential exit may be tangent, but it cannot be on the straight extension of the previous path.

1085 'Helical path programmed incorrectly'

DETECTED During execution.

CAUSE When programming an arc with the format: "G02/G03 X Y I J Z K" the programmed helical path cannot be carried out. The desired height cannot be reached with the programmed helical pitch.

SOLUTION Correct the syntax of the block. The height of the interpolation and the coordinates of the end point in the plane must be related taking the helical pitch into consideration.

1086 'The Spindle cannot be referenced (homed)'

CAUSE Spindle machine parameter REFEED1(P34) is set to «0».

1087 'Circle with zero radius'

DETECTED During execution.

CAUSE/S The probable causes might be:
1.- When programming an arc with the format: "G02/G03 X Z I K", a circular interpolation has been programmed with «zero» radius.
2.- While working with tool radius compensation, an inside arc has been programmed with a radius equal to the tool radius.

SOLUTION The solution for each cause is:
1.- Arcs with zero radius cannot be programmed. Program a radius value other than zero.
2.- When working with tool radius compensation, the arc radius must be greater than the tool radius. Otherwise, the tool cannot machine the programmed path because the tool would have to machine an arc with zero radius.

1088 'Zero offset range exceeded'

DETECTED During execution.

CAUSE A zero offset has been programmed and the end position has too high a value.

SOLUTION Check that the values assigned to the zero offsets (G54-G59) are correct. If the offset values have been assigned from a program using parameters, check that the parameter values are correct. If an absolute zero offset (G54-G57) has been programmed and an incremental one (G58-G59), check that the sum of both does not exceed the travel limits of the machine.

1089 'Work zone limit range exceeded'

DETECTED During execution.

CAUSE Work zone limits "G20" or "G21" have been programmed using parameters and the value of the parameter is greater than the one allowed for this function.

SOLUTION Check the program history so this parameter does not reach with that value to the block defining those limits.

1090 'Point within the forbidden zone 1'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 1 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 1 (defined with G20/G21) has been defined as «no entry zone» (G22 K1 S1). To disable it, program "G22 K1 S0".

1091 'Point within the forbidden zone 2'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 2 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 2 (defined with G20/G21) has been defined as «no entry zone» (G22 K2 S1). To disable it, program "G22 K2 S0".

1092 'Insufficient accelerations for the programmed threadcutting feedrate'

DETECTED During execution.

CAUSE A threading operation has been programmed with not enough room to accelerate and decelerate.

SOLUTION Program a lower feedrate.

1096 'Point within the forbidden zone 3'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 3 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 3 (defined with G20/G21) has been defined as «no entry zone» (G22 K3 S1). To disable it, program "G22 K3 S0".

1097 'Point within the forbidden zone 4'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 4 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 4 (defined with G20/G21) has been defined as «no entry zone» (G22 K4 S1). To disable it, program "G22 K4 S0".

1098 'Wrong work zone boundaries'

DETECTED During execution.

CAUSE The upper limits (G21) of the work zone defined are equal to or less than its lower limits (G20)

SOLUTION The upper limits (G21) of the work zone must always be greater than its lower limits (G20).

1099 'Do not program a slaved axis'

DETECTED During execution.

CAUSE While working with polar coordinates, a movement has been programmed which implies moving an axis which is slaved to another one.

SOLUTION The movements in polar coordinates are carried out on the main axes of the work plane. Therefore, the axes defining a plane cannot be slaved to each other or to a third axis. To free the axes, program "G78".

1100 'Spindle travel limit overrun'

DETECTED During execution.

CAUSE An attempt has been made to exceed the physical travel limits of the spindle. Consequently, the PLC activates the spindle marks: "LIMIT+S" or "LIMIT-S" ("LIMIT+S2" or "LIMIT-S2" when working with the second spindle)

1101 'Spindle locked'

DETECTED During execution.

CAUSE The CNC tries to output the analog voltage to the drive while the spindle input SERVOSON is still low. The error may come up due to an error in the PLC program where this signal is treated wrong or maybe the value of the spindle parameter DWELL(P17) is too low.

1102 'Spindle following error limit overrun'

DETECTED During execution.

CAUSE While the spindle is operating in closed loop (M19), its following error is greater than the values indicated by spindle parameters MAXFLWE1(P21) or MAXFLE2(P22). The probable causes for this error are:

DRIVE FAILURE
- Defective drive.
- Enable signals missing.
- Power supply missing.
- Poor drive adjustment.
- Velocity command signal missing.

MOTOR FAILURE
- Defective motor.
- Power wiring.

FEEDBACK FAILURE
- Defective feedback device.
- Defective feedback cable.

CNC FAILURE
- Defective CNC.
- Wrong parameter setting.

MECHANICAL FAILURE
- Mechanical friction.
- Spindle mechanically locked up

1110-1118 '* axis range exceeded'

DETECTED During execution.

CAUSE A movement has been defined using parameters and the value of the parameter is greater than the maximum axis travel allowed.

SOLUTION Check the history of the program so that parameter does not reach with that value to the block where that movement has been programmed.

1119-1127 '* axis cannot be synchronized'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- An attempt has been made to synchronize an axis with another one from the PLC, but the axis is already slaved to another one with function "G77".

2.- When programming or trying to move an axis already synchronized with another one.

1128-1136 '* axis maximum feed exceeded'

DETECTED During execution.

CAUSE The resulting feedrate of some axis after applying the particular scaling factor exceeds the maximum value indicated by axis machine parameter MAXFEED (P42).

1137-1145 'Incorrect * axis feedrate parameter'

DETECTED During execution.

CAUSE "G00" has been programmed with axis parameter G00FEED(P38)=0 or "G1 F00" has been programmed with axis machine parameter MAXFEED(P42) = 0.

1146-1154 '* axis locked'

DETECTED During execution.

CAUSE The CNC tries to output the velocity command to the drive while the spindle input SERVO(n)ON is still low. The error may come up due to an error in the PLC program where this signal is treated wrong or maybe the value of the spindle parameter DWELL(P17) is too low.

1155-1163 '* axis soft limit overrun'

DETECTED During execution.

CAUSE A coordinate has been programmed which is beyond the limits defined by axis machine parameters LIMIT+(P5) and LIMIT-(P6).

1164-1172 '* axis work zone 1 overrun'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 1 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 1 (defined with G20/G21) has been defined as «no exit zone» (G22 K1 S2). To disable it, program "G22 K1 S0".

1173-1181 '* axis work zone 2 overrun'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 2 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 2 (defined with G20/G21) has been defined as «no exit zone» (G22 K2 S2). To disable it, program "G22 K2 S0".

1182-1190 '* axis following error limit overrun'

DETECTED During execution.

CAUSE The following error of the axis is greater than the values indicated by spindle parameters MAXFLWE1(P21) or MAXFLE2(P22). The probable causes for this error are:

DRIVE FAILURE
- Defective drive.
- Enable signals missing.
- Power supply missing.
- Poor drive adjustment.
- Velocity command signal missing.

MOTOR FAILURE
- Defective motor.
- Power wiring.

FEEDBACK FAILURE
- Defective feedback device.
- Defective feedback cable.

CNC FAILURE
- Defective CNC.
- Wrong parameter setting.

MECHANICAL FAILURE
- Mechanical friction.
- Axis mechanically locked up

1191-1199 'Coupled * axis following error difference too large'

CAUSE The «n» axis is electronically coupled to another one or it is slaved to a Gantry axis and the difference between their following errors is greater than the value set by axis machine parameter MAXCOUPE(P45).

1200-1208 '* axis hard limit overrun'

DETECTED During execution.

CAUSE An attempt has been made to exceed the physical travel limits of the axis. Consequently, the PLC activates the axis marks: "LIMIT+1" or "LIMIT-1"

1209-1217 '* axis servo error'

CAUSE The actual axis speed, after a time period indicated by axis machine parameter FBALTIME(P12), is below 50% or over 200% of the programmed value.

1218-1226 '* axis work zone 3 overrun'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 3 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 3 (defined with G20/G21) has been defined as «no exit zone» (G22 K3 S2). To disable it, program "G22 K3 S0".

1228-1236 '* axis work zone 4 overrun'

DETECTED During execution

CAUSE An attempt has been made to move an axis to a point located inside the work zone 4 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 4 (defined with G20/G21) has been defined as «no exit zone» (G22 K4 S2). To disable it, program "G22 K4 S0".

1237 'Do not change entry angle inside the thread'

DETECTED While executing.

CAUSE A multiple thread has been defined, but an entry angle "Q" has been programmed between two threads.

SOLUTION When making multiple threads, only the first one may have the entry angle "Q" .

HARDWARE ERRORS

2000 'External emergency activated'

DETECTED During execution.

CAUSE PLC input I1 has been set to zero (possible E-stop button) or the PLC mark M5000(/EMERGEN) has been set to zero.

SOLUTION Check at the PLC why these inputs are set to zero. (Maybe power is missing).

2001-2009 '* axis feedback error'

DETECTED During execution.

CAUSE The CNC does not receive feedback signals from the axes.

SOLUTION Check the feedback connections.

NOTE:This error comes up on differential feedback signals (double-ended signals), DIFFBACK(P9)=YES, and sinewave feedback signals SINMAGNI(P10) other than zero, when parameter FBACKAL(P11)=ON. This error can be avoided by setting parameter FBACKAL(P11)=OFF, although this solution is only temporary.

2010 'Spindle feedback error'

DETECTED During execution.

CAUSE The CNC does not receive the spindle feedback signals.

SOLUTION Check the feedback connections.

NOTE:This error comes up on differential feedback signals (double-ended signals), DIFFBACK(P14)=YES, when parameter FBACKAL(P15)=ON. This error can be avoided by setting parameter FBACKAL(P15)=OFF, although this solution is only temporary.

2011 'Maximum temperature exceeded'

DETECTED Any time.

CAUSE The maximum internal CNC temperature exceeded. The probable causes might be:

- Poor ventilation of the electrical cabinet (enclosure).
- Axis board with some defective component.

SOLUTION Turn the CNC off and wait until it cools off. If the error persists, some component of the board may be defective. In that case, contact the Service Department to replace the board.

2012 'Axes board without voltage'

DETECTED During execution.

CAUSE The 24V are missing from the outputs of the axes board. The fuse might be blown.

SOLUTION Supply the outputs of the axes board with 24V. If the fuse is blown, replace it.

2013 'I/O 1 board without voltage'
2014 'I/O 2 board without voltage'
2015 'I/O 3 board without voltage'

DETECTED During execution.

CAUSE The 24V are missing from the outputs of the corresponding I/O board. The fuse might be blown.

SOLUTION Supply the outputs of the corresponding I/O board with 24V. If the fuse is blown, replace it.

2016 'PLC not ready.'

DETECTED During execution.

CAUSE The PLC program is not running. The probable causes might be:
- There is no PLC program
- WATCHDOG error
- The program has been stopped from the monitoring mode.

SOLUTION Restart the PLC program by restarting the PLC.

2017 'CNC RAM memory error'

DETECTED While starting the CNC up or during diagnosis.

CAUSE A RAM memory problem has been detected at the CNC.

SOLUTION Change the CPU board. Contact the Service Department.

2018 'CNC EPROM memory error'

DETECTED While starting the CNC up or during diagnosis..

CAUSE An EPROM memory problem has been detected at the CNC.

SOLUTION Change the EPROM. Contact the Service Department.

2019 'PLC RAM memory error'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A RAM memory problem has been detected at the PLC.

SOLUTION Change the PLC board. Contact the Service Department.

2020 'PLC EPROM memory error'

DETECTED While starting the CNC up or during diagnosis..

CAUSE An EPROM memory problem has been detected at the PLC.

SOLUTION Change the EPROM. Contact the Service Department.

2021 'USER RAM memory error at the CNC. Press any key.'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A user RAM memory problem has been detected at the CNC.

SOLUTION Contact the Service Department.

2022 'CNC system RAM memory error. Press any key.'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A system RAM memory problem has been detected at the CNC.

SOLUTION Contact the Service Department.

2023 'PLC RAM error. Press any key.'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A RAM memory problem has been detected at the PLC.

SOLUTION Contact the Service Department.

2024 'The tracing module has no voltage'

DETECTED During execution.
CAUSE The 24V are missing from the outputs of the tracing board. The fuse might be blown.
SOLUTION Supply the outputs of the tracing board with 24V. If the fuse is blown, replace it.

2026 'Maximum probe travel overrun'

DETECTED During execution.
CAUSE The probe has exceeded the maximum deflection allowed by machine parameter.
SOLUTION Reduce the feedrate and check that the probe is not damaged.

2027 'SERCOS chip RAM Error. Press a key.'

DETECTED While starting the CNC up or during diagnosis..
CAUSE A RAM memory problem has been detected at the SERCOS chip.
SOLUTION Change the SERCOS board. Contact the Service Department.

2028 'SERCOS chip version Error. Press a key.'

DETECTED While starting the CNC up.
CAUSE The SERCOS chip version is old.
SOLUTION Change the SERCOS chip. Contact the Service Department.

PLC ERRORS

3000 '(PLC_ERR without description)'

DETECTED During execution.

CAUSE Marks ERR1 through ERR64 have been set to "1".

SOLUTION Check why these marks are set to "1" in the PLC program and act accordingly.

3001 'WATCHDOG in Main Module (PRG).'

DETECTED At any time.

CAUSE/S The probable causes might be:
1.- The main PLC program execution takes longer than the time period set by PLC parameter WAGPRG(P0).
2.- The program is in a loop.

SOLUTION Increase the time period of PLC parameter WAGPRG(P0) or increase the PLC processing speed.
- Insert the CPU TURBO.
- Change the PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3002 'WATCHDOG in Periodic Module (PE).'

DETECTED At any time.

CAUSE/S The probable causes might be:
1.- The periodic PLC program execution takes longer than the time period set by PLC parameter WAGPER(P1).
2.- The program is in a loop.

SOLUTION Increase the time period of PLC parameter WAGPER(P1) or increase the PLC processing speed.
- Insert the CPU TURBO.
- Change the PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3003 'Division by zero in PLC.'

DETECTED At any time.

CAUSE The PLC program contains a line whose execution involves a division by zero.

SOLUTION When working with registers, that register may have receive the zero value throughout the program history. Check that the register does not reach the operation with that value.

3004 'PLC Error -> '

DETECTED At any time.

CAUSE An error has been detected on the PLC board.

SOLUTION Change the PLC board. Contact the Service Department.

DRIVE ERRORS

4000 'SERCOS ring error'

DETECTED During execution.

CAUSE SERCOS communication has been interrupted. This could be because there has been an interruption in the connection ring (disconnected or broken fiber link) or the wrong configuration:

1.- The node selector switch position does not match the sercosid.
2.- Parameter P120 (SERSPD) does not match the transmission speed.
3.- The drive version is not compatible with the CNC.
4.- An error has been detected on the SERCOS board.
5.- The transmission speeds are different at the drive and at the CNC.

SOLUTION To check that the connection ring has not been interrupted, verify that the light travels through the optical fiber. If it is due to the wrong configuration, contact the Service Department.

4002 'Drive overload (201)'
4003 'Drive overtemperature (107)'
4004 'Motor overtemperature (108)'
4005 'Heat-sink overtemperature (106)'
4006 'Voltage control error (100...105)'
4007 'Feedback error (600...606)'
4008 'Power bus error (213...215)'
4009 'Overcurrent (212)'
4010 'Power bus overvoltage (304/306)'
4011 'Power bus undervoltage (307)'

DETECTED During execution.

CAUSE An error has been detected at the drive. The number in brackets indicates the standard error number of the drive. Refer to the drive manual for further information.

SOLUTION These types of errors come with messages 4019, 4021, 4022 or 4023 which indicate at which axis drive or spindle drive the error has come up. Refer to the drive manual for the error (number in brackets) and act accordingly.

4016 'Error, undefined class 1'

DETECTED During execution.

CAUSE The drive has detected an error, but it cannot identify it.

SOLUTION Contact the Service Department.

4017 'Drive error'

DETECTED During execution.

CAUSE An error has been detected at the drive which does not match the standard SERCOS errors.

SOLUTION These types of errors come with messages 4019, 4021, 4022 or 4023 which indicate at which axis drive or spindle drive the error has come up. Refer to the drive manual for the error and act accordingly.

4018 'Sercos variable accessing error'

DETECTED During execution.

CAUSE An attempt has been made to read (or write) a SERCOS variable from the CNC, but:
1.- The variable does not exist.
2.- The maximum/minimum values have been exceeded.
3.- The SERCOS variable has variable length
4.- the variable is read-only and cannot be written.

SOLUTION Check that the variable is of the right type for that particular action.

4019 'Axis drive error on: '

DETECTED During execution.

CAUSE These messages come with errors 4002 - 4011. When one of those errors come up, it indicates on which axis it came up.

4021 'Spindle drive error'
4022 '2nd spindle drive error'
4023 'Auxiliary spindle drive error'

DETECTED During execution.

CAUSE These messages come with errors 4002 - 4011. When one of those errors come up, it indicates on which spindle it came up.

4024 'SERCOS error when homing'

DETECTED During execution.

CAUSE The SERCOS home searching command has been executed wrong.

4025 'SERCOS ring error 1'

DETECTED During execution.

CAUSE The time it takes to calculate the axis speed exceeds the cycle time set to transmit to the drive.

SOLUTION Contact the Service Department.

TABLE DATA ERRORS

echk_gen 'CHECKSUM ERROR: GENERAL PARAMETERS Initialize? (ENTER/ESC)'
echk_cab 'CHECKSUM ERROR: SPINDLE PARAMETERS Initialize? (ENTER/ESC)'
echk_cab2 'CHECKSUM ERROR:2nd SPINDLE PARAMETERS Initialize? (ENTER/ESC)'
echk_cax 'CHECKSUM ERROR:AUX.SPINDLE PARAMETERS Initialize? (ENTER/ESC)'
echk_rs1 'CHECKSUM ERROR:SERIAL LINE 1 PARAMETERS Initialize? (ENTER/ESC)'
echk_rs2 'CHECKSUM ERROR:SERIAL LINE 2 PARAMETERS Initialize? (ENTER/ESC)'
echk_plc 'CHECKSUM ERROR:PLC PARAMETERS Initialize? (ENTER/ESC)'

DETECTED While starting the CNC up.

CAUSE Data lost in the tables. Possible RAM error.

SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

echk_org 'CHECKSUM ERROR:ZERO OFFSET TABLE Initialize? (ENTER/ESC)'
echk_psw 'CHECKSUM ERROR:PASSWORD TABLE Initialize? (ENTER/ESC)'

DETECTED While starting the CNC up.

CAUSE Data lost in the tables. Possible RAM error.

SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

echk_ejex 'CHECKSUM ERROR:AXIS X PARAMETERS Initialize? (ENTER/ESC)'
echk_ejey 'CHECKSUM ERROR:AXIS Y PARAMETERS Initialize? (ENTER/ESC)'
echk_ejez 'CHECKSUM ERROR:AXIS Z PARAMETERS Initialize? (ENTER/ESC)'
echk_ejeu 'CHECKSUM ERROR:AXIS U PARAMETERS Initialize? (ENTER/ESC)'
echk_ejev 'CHECKSUM ERROR:AXIS V PARAMETERS Initialize? (ENTER/ESC)'
echk_ejew 'CHECKSUM ERROR:AXIS W PARAMETERS Initialize? (ENTER/ESC)'
echk_ejea 'CHECKSUM ERROR:AXIS A PARAMETERS Initialize? (ENTER/ESC)'
echk_ejeb 'CHECKSUM ERROR:AXIS B PARAMETERS Initialize? (ENTER/ESC)'
echk_ejec 'CHECKSUM ERROR:AXIS C PARAMETERS Initialize? (ENTER/ESC)'

DETECTED While starting the CNC up.

CAUSE Data lost in the axis parameter tables. Possible RAM error.

SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

echk_herr ‘CHECKSUM ERROR:TOOL TABLE Initialize? (ENTER/ESC)''
echk_corr ‘CHECKSUM ERROR:TOOL OFFSET TABLE Initialize? (ENTER/ESC)’
echk_alm ‘CHECKSUM ERROR:MAGAZINE TABLE Initialize? (ENTER/ESC)’
echk_aux ‘CHECKSUM ERROR:M FUNCTION TABLE Initialize? (ENTER/ESC)’
echk_husx ‘CHECKSUM ERROR:LEADSCREW X TABLE Initialize? (ENTER/ESC)’
echk_husy ‘CHECKSUM ERROR:LEADSCREW Y TABLE Initialize? (ENTER/ESC)’
echk_husz ‘CHECKSUM ERROR:LEADSCREW Z TABLE Initialize? (ENTER/ESC)’
echk_husu ‘CHECKSUM ERROR:LEADSCREW U TABLE Initialize? (ENTER/ESC)’
echk_husv ‘CHECKSUM ERROR:LEADSCREW V TABLE Initialize? (ENTER/ESC)’
echk_husw ‘CHECKSUM ERROR:LEADSCREW W TABLE Initialize? (ENTER/ESC)’
echk_husa ‘CHECKSUM ERROR:LEADSCREW A TABLE Initialize? (ENTER/ESC)’
echk_husb ‘CHECKSUM ERROR:LEADSCREW B TABLE Initialize? (ENTER/ESC)’
echk_husc ‘CHECKSUM ERROR:LEADSCREW C TABLE Initialize? (ENTER/ESC)’
echk_cru1 ‘CHECKSUM ERROR:CROSS COMP. TABLE 1 Initialize? (ENTER/ESC)’
echk_cru2 ‘CHECKSUM ERROR:CROSS COMP. TABLE 2 Initialize? (ENTER/ESC)’
echk_cru3 ‘CHECKSUM ERROR:CROSS COMP. TABLE 3 Initialize? (ENTER/ESC)’

DETECTED While starting the CNC up.

CAUSE Data lost in the tables. Possible RAM error.

SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

eincx ‘Incorrect X axis leadscrew table. Press any key’
eincy ‘Incorrect Y axis leadscrew table. Press any key’
eincz ‘Incorrect Z axis leadscrew table. Press any key’
eincu ‘Incorrect U axis leadscrew table. Press any key’
eincv ‘Incorrect V axis leadscrew table. Press any key’
eincw ‘Incorrect W axis leadscrew table. Press any key’
einca ‘Incorrect A axis leadscrew table. Press any key’
eincb ‘Incorrect B axis leadscrew table. Press any key’
eincc ‘Incorrect C axis leadscrew table. Press any key’

DETECTED While starting the CNC up.

CAUSE Wrong data in the leadscrew compensation table.

SOLUTION The points must be defined in the table as follows:
- They must be ordered according to their position on the axis starting from the most negative or least positive point to be compensated for.
- The machine reference point must have an error value of zero.
- The error difference between two points cannot be greater than the distance between them.

einx1 ‘Incorrect cross compensation table 1’
einx2 ‘Incorrect cross compensation table 2’
einx3 ‘Incorrect cross compensation table 3’

DETECTED While starting the CNC up.

CAUSE Wrong data in the cross compensation table.

SOLUTION The points must be defined in the table as follows:
- They must be ordered according to their position on the axis starting from the most negative or least positive point to be compensated for.
- The machine reference point must have an error value of zero.

einxx 'Incorrect cross compensation table parameters'

DETECTED While starting the CNC up.

CAUSE The parameters indicating which axis take part in the cross compensation are defined wrong.

SOLUTION A nonexistent axis might have been defined or that the axis affected by the compensation is the same as the one causing the error.

esercos 'Wrong sercosid parameters for axes and spindle'

DETECTED While starting the CNC up.

CAUSE The sercosid parameters are wrong.

SOLUTION The sercosid parameters—:

- must start from 1.
- must be consecutive.
- must not be repeated. (Except on lathes with a "C" axis. The spindle and the "C" axis may share the same sercosid).

ERRORS IN 8055TC OPERATING MODE

Errors in the turning cycle.

'Cycle without roughing or finishing'
DETECTED While executing.
CAUSE No tool has been selected for the roughing or finishing operations.
SOLUTION Select the tool for roughing (If T=0 there is no roughing) and for finishing (If T=0 there is no finishing).

'ROUGHING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'ROUGHING: Value of F=0'
DETECTED While executing.
CAUSE The roughing feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'ROUGHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for roughing.
SOLUTION Program a positive spindle speed «S» other than zero.

'FINISHING: Value of F=0'
DETECTED While executing.
CAUSE No feedrate «F» has been defined for finishing.
SOLUTION Program a positive feedrate other than zero.

'FINISHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for finishing.
SOLUTION Program a positive spindle speed «S» other than zero.

'GEOMETRY: Value of Zi=Zf'
DETECTED While executing.
CAUSE The Z coordinates of the starting and end points are the same.
SOLUTION The Z coordinates of the starting and end points must be different.

'GEOMETRY: Value of X=Ø'
DETECTED While executing.
CAUSE The coordinates of the starting and end diameters are the same.
SOLUTION The X coordinates of the starting and end points must be different from the end diameter.

'FINISHING: Wrong stock'
DETECTED While executing.
CAUSE The finishing stock is greater than the total machining depth.
SOLUTION The finishing stock must be smaller than the total machining depth

'GEOMETRY: Final diameter is not external'
DETECTED While executing.
CAUSE In an outside diameter, the final diameter is greater than the initial one.
SOLUTION In an outside diameter, the final diameter must be smaller than the initial one.

'GEOMETRY: Final diameter is not internal'
DETECTED While executing.
CAUSE In an inside diameter, the final diameter is smaller than the initial one
SOLUTION In an inside diameter, the final diameter must be greater than the initial one.

Errors in the facing cycle.

'Cycle without roughing or finishing'
DETECTED While executing.
CAUSE No tool has been selected for roughing or finishing.
SOLUTION Select the tool for roughing (If T=0 there is no roughing) and for finishing (If T=0 there is no finishing).

'ROUGHING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'ROUGHING: Value of F=0'
DETECTED While executing.
CAUSE The roughing feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'ROUGHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for roughing.
SOLUTION Program a positive spindle speed «S» other than zero.

'FINISHING: Value of F=0'
DETECTED While executing.
CAUSE No feedrate «F» has been defined for finishing.
SOLUTION Program a positive feedrate other than zero.

FINISHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for finishing.
SOLUTION Program a positive spindle speed «S» other than zero.

'GEOMETRY: Value of Zi=Zf'
DETECTED While executing.
CAUSE The Z coordinates of the starting and end points are the same.
SOLUTION The Z coordinates of the starting and end points must be different.

'GEOMETRY: Value of X=Ø'
DETECTED While executing.
CAUSE The coordinates of the starting and end diameters are the same.
SOLUTION The X coordinates of the starting and end diameters must be different from the end diameter.

'FINISHING: Wrong stock'
DETECTED While executing.
CAUSE The finishing stock is greater than the total machining depth.
SOLUTION The finishing stock must be smaller than the total machining depth

Errors in taper cycles.

'Cycle without roughing or finishing'
DETECTED While executing.
CAUSE No tool has been selected for roughing or finishing.
SOLUTION Select the tool for roughing (If T=0 there is no roughing) and for finishing (If T=0 there is no finishing).

'ROUGHING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'ROUGHING: Value of F=0'
DETECTED While executing.
CAUSE The roughing feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'ROUGHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for roughing.
SOLUTION Program a positive spindle speed «S» other than zero.

'FINISHING: Value of F=0'
DETECTED While executing.
CAUSE No feedrate «F» has been defined for finishing.
SOLUTION Program a positive feedrate other than zero.

'FINISHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for finishing.
SOLUTION Program a positive spindle speed «S» other than zero.

'GEOMETRY: Value of Zi=Zf'
DETECTED While executing.
CAUSE The Z coordinates of the starting and end points are the same.
SOLUTION The Z coordinates of the starting and end points must be different.

'GEOMETRY: Value of X=Ø'
DETECTED While executing.
CAUSE The coordinates of the starting and end diameters are the same.
SOLUTION The X coordinates of the starting and end diameters must be different from the end diameter.

'GEOMETRY: Wrong angle value'
DETECTED While executing.
CAUSE The taper angle is smaller than 0º or greater than 90º.
SOLUTION The taper angle must be in the 0º to 90º range.

'GEOMETRY: Wrong Quadrant'
DETECTED While executing.
CAUSE A taper has been defined in the wrong quadrant.
SOLUTION Select the right quadrant with the corresponding icon.

'No negative safety distance permitted in this cycle'
DETECTED While executing.
CAUSE A negative safety distance has been defined.
SOLUTION Taper canned cycles require a positive safety distance.

Errors in the rounding cycles.

'Cycle without roughing or finishing'
DETECTED While executing.
CAUSE No tool has been selected for roughing or finishing.
SOLUTION Select the tool for roughing (If T=0 there is no roughing) and for finishing (If T=0 there is no finishing).

'ROUGHING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'ROUGHING: Value of F=0'
DETECTED While executing.
CAUSE The roughing feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'ROUGHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for roughing.
SOLUTION Program a positive spindle speed «S» other than zero.

'FINISHING: Value of F=0'
DETECTED While executing.
CAUSE No feedrate «F» has been defined for finishing.
SOLUTION Program a positive feedrate other than zero.

'FINISHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for finishing.
SOLUTION Program a positive spindle speed «S» other than zero.

'GEOMETRY: Wrong radius value'
DETECTED While executing.
CAUSE The rounding radius has not been defined.
SOLUTION Program a rounding radius other than zero.

'No negative safety distance permitted in this cycle'
DETECTED While executing.
CAUSE A negative safety distance has been defined.
SOLUTION Rounding canned cycles require a positive safety distance.

Errors in the threading cycle.

'GEOMETRY: Value of Zi=Zf'
DETECTED While executing.
CAUSE The Z coordinates of the starting and end points are the same.
SOLUTION The Z coordinates of the starting and end points must be different.

'THREADING: Value of T=0'
DETECTED While executing.
CAUSE No tool number has been defined.
SOLUTION The tool number must be other than zero.

'THREADING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been programmed.
SOLUTION Program a positive spindle speed «S» other than zero.

'THREADING: Value of P=0'
DETECTED While executing.
CAUSE The thread pitch has not been programmed.
SOLUTION Program a thread pitch greater than zero.

'THREADING: Value of H=0'
DETECTED While executing.
CAUSE The depth of the thread has not been defined.
SOLUTION Program a thread depth other than zero.

'THREADING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'THREADING: Value of σ > (Zf-Zi)'
DETECTED While executing.
CAUSE The distance to the end of the thread is greater than its length.
SOLUTION Program a distance to the end of the thread smaller than its length.

'GEOMETRY: Value of Xi=Xf'
DETECTED While executing.
CAUSE The X coordinates of the starting and end points are the same.
SOLUTION The X coordinates of the starting and end points must be different.

'THREADING: Value of σ > (Xf-Xi)'
DETECTED While executing.
CAUSE The distance to the end of the thread is greater than its length.
SOLUTION Program a distance to the end of the thread smaller than its length.

Errors in the grooving cycles.

'Cycle without roughing or finishing'
DETECTED While executing.
CAUSE No tool has been selected for roughing or finishing.
SOLUTION Select the tool for roughing (If T=0 there is no roughing) and for finishing (If T=0 there is no finishing).

'ROUGHING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'ROUGHING: Value of F=0'
DETECTED While executing.
CAUSE The roughing feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'ROUGHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for roughing.
SOLUTION Program a positive spindle speed «S» other than zero.

'FINISHING: Value of F=0'
DETECTED While executing.
CAUSE No feedrate «F» has been defined for finishing.
SOLUTION Program a positive feedrate other than zero.

'FINISHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for finishing.
SOLUTION Program a positive spindle speed «S» other than zero.

'GEOMETRY: Value of Zi=Zf'
DETECTED While executing.
CAUSE The Z coordinates of the starting and end points are the same.
SOLUTION The Z coordinates of the starting and end points must be different.

'GEOMETRY: Value of X=Ø'
DETECTED While executing.
CAUSE The coordinates of the starting and end diameters are the same.
SOLUTION The X coordinates of the starting and end diameters must be different from the end diameter.

'FINISHING: Wrong stock'
DETECTED While executing.
CAUSE The finishing stock is greater than the total machining depth.
SOLUTION The finishing stock must be smaller than the total machining depth

'ROUGHING: Wrong tool shape code'
DETECTED While executing.
CAUSE The roughing of the groove cannot be done with the selected location code (tool shape code).
SOLUTION Select a tool with the right location code (shape).

'FINISHING: Wrong tool shape code'
DETECTED While executing.
CAUSE The finishing of the groove cannot be done with the selected location code (tool shape code).
SOLUTION Select a tool with the right location code (shape).

'GEOMETRY: Final diameter is not external'
DETECTED While executing.
CAUSE An outside groove has been defined with a final diameter larger than the initial one.
SOLUTION In an outside groove, the final diameter must be smaller than the initial one.

'GEOMETRY: Final diameter is not internal'
DETECTED While executing.
CAUSE An inside groove has been defined with a final diameter smaller than the initial one.
SOLUTION In an inside groove, the final diameter must be larger than the initial one.

'ROUGHING: Wrong tool for GROOVING.'
DETECTED While executing.
CAUSE The selected tool has the wrong geometry for this operation.
SOLUTION Select a tool with the right geometry

'FINISHING: Wrong tool for GROOVING.'
DETECTED While executing.
CAUSE The selected tool has the wrong geometry for this operation.
SOLUTION Select a tool with the right geometry

'GEOMETRY: Wrong angle for GROOVING.'
DETECTED While executing.
CAUSE The angle of the groove walls is either smaller than 0º or greater than 90º.
SOLUTION The angle of the groove walls must be in the 0º to 90º range.

'GEOMETRY: The sides of the groove cut each other.'
DETECTED While executing.
CAUSE The two walls of the groove intersect each other.
SOLUTION Check the cycle data. The groove walls must not intersect each other.

Errors in the profile cycles.

'Cycle without roughing or finishing'
DETECTED While executing.
CAUSE No tool has been selected for roughing or finishing.
SOLUTION Select the tool for roughing (If T=0 there is no roughing) and for finishing (If T=0 there is no finishing).

'ROUGHING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'ROUGHING: Value of F=0'
DETECTED While executing.
CAUSE The roughing feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'ROUGHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for roughing.
SOLUTION Program a positive spindle speed «S» other than zero.

'FINISHING: Value of F=0'
DETECTED While executing.
CAUSE No feedrate «F» has been defined for finishing.
SOLUTION Program a positive feedrate other than zero.

'FINISHING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been defined for finishing.
SOLUTION Program a positive spindle speed «S» other than zero.

'No negative safety distance permitted in this cycle'
DETECTED While executing.
CAUSE A negative safety distance has been defined.
SOLUTION The profile canned cycles require a positive safety distance.

Errors in the profile cycles of the «C» axis.

'PROFILE CYCLE «C» AXIS: Value of T=0'
DETECTED While executing.
CAUSE No tool number has been defined.
SOLUTION The tool number must be other than zero.

'PROFILE CYCLE «C» AXIS: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'PROFILE CYCLE «C» AXIS: Value of I=0'
DETECTED While executing.
CAUSE The total machining depth has not been defined.
SOLUTION Program a machining depth other than zero.

'PROFILE CYCLE «C» AXIS: Value of F=0'
DETECTED While executing.
CAUSE The feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'PROFILE CYCLE «C» AXIS: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «St» of the live tool has not been defined.
SOLUTION Program a positive spindle speed «St» other than zero.

Errors in the drilling cycle.

'DRILLING: Value of T=0'
DETECTED While executing.
CAUSE No tool number has been defined.
SOLUTION The tool number must be other than zero.

'DRILLING: Value of Δ=0'
DETECTED While executing.
CAUSE The cutting depth (pass) has not been defined «Δ».
SOLUTION Program a pass greater than zero.

'DRILLING: Value of L=0'
DETECTED While executing.
CAUSE The drilling depth has not been defined.
SOLUTION Program a drilling depth other than zero..

'DRILLING: Value of F=0'
DETECTED While executing.
CAUSE The feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'DRILLING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been programmed.
SOLUTION Program a positive spindle speed «S» other than zero.

Errors in the multiple drilling cycle.

'DRILLING: Value of T=0'
DETECTED While executing.
CAUSE No tool number has been defined.
SOLUTION The tool number must be other than zero.

'DRILLING: Value of Δ=0'
DETECTED While executing.
CAUSE The drilling peck has not been defined «Δ».
SOLUTION Program a drilling peck greater than zero.

'DRILLING: Value of L=0'
DETECTED While executing.
CAUSE The drilling depth has not been defined.
SOLUTION Program a drilling depth other than zero..

'DRILLING: Value of F=0'
DETECTED While executing.
CAUSE The feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'DRILLING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «St» of the live tool has not been defined.
SOLUTION Program a positive spindle speed «St» other than zero.

'MULTIPLE CYCLE: Value of β=0'
DETECTED While executing.
CAUSE The angular step between operations has not been programmed.
SOLUTION Program an angular step other than zero.

'MULTIPLE CYCLE: Value of N=0'
DETECTED While executing.
CAUSE The number of operations has not been defined.
SOLUTION The minimum number of machining operations is «1».

Errors in the tapping cycle.

'TAPPING: Value of T=0'
DETECTED While executing.
CAUSE No tool number has been defined.
SOLUTION The tool number must be other than zero.

'TAPPING: Value of L=0'
DETECTED While executing.
CAUSE The tapping depth has not been defined.
SOLUTION Program a tapping depth other than zero.

'TAPPING: Value of F=0'
DETECTED While executing.
CAUSE The feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'TAPPING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «S» has not been programmed.
SOLUTION Program a positive spindle speed «S» other than zero.

Errors in the multiple tapping cycle.

'TAPPING: Value of T=0'
DETECTED While executing.
CAUSE No tool number has been defined.
SOLUTION The tool number must be other than zero.

'TAPPING: Value of L=0'
DETECTED While executing.
CAUSE The tapping depth has not been defined.
SOLUTION Program a tapping depth other than zero.

'TAPPING: Value of F=0'
DETECTED While executing.
CAUSE The feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'TAPPING: Value of S=0'
DETECTED While executing.
CAUSE The spindle speed «St» of the live tool has not been defined.
SOLUTION Program a positive spindle speed «St» other than zero.

'MULTIPLE CYCLE: Value of β=0'
DETECTED While executing.
CAUSE The angular step between operations has not been programmed.
SOLUTION Program an angular step other than zero.

'MULTIPLE CYCLE: Value of N=0'
DETECTED While executing.
CAUSE The number of operations has not been defined.
SOLUTION The minimum number of machining operations is «1».

Errors in the multiple slot milling cycle.

'MULTIPLE CYCLE: Value of β=0'

DETECTED While executing.
CAUSE The angular step between operations has not been programmed.
SOLUTION Program an angular step other than zero.

'MULTIPLE CYCLE: Value of N=0'

DETECTED While executing.
CAUSE The number of operations has not been defined.
SOLUTION The minimum number of machining operations is «1».

'MULTIPLE SLOT MILLING CYCLE: Value of T=0'

DETECTED While executing.
CAUSE No tool number has been defined.
SOLUTION The tool number must be other than zero.

'MULTIPLE SLOT MILLING CYCLE: Value of F=0'

DETECTED While executing.
CAUSE The feedrate «F» has not been defined.
SOLUTION Program a positive feedrate other than zero.

'MULTIPLE SLOT MILLING CYCLE: Value of S=0'

DETECTED While executing.
CAUSE The spindle speed «St» of the live tool has not been defined.
SOLUTION Program a positive spindle speed «St» other than zero.

'MULTIPLE SLOT MILLING CYCLE: Value of I=0'

DETECTED While executing.
CAUSE The depth of the slot has not been defined.
SOLUTION Program a depth of the slot other than zero.

'MULTIPLE SLOT MILLING CYCLE: Value of L=0'

DETECTED While executing.
CAUSE The length of the slot has not been defined.
SOLUTION The length of the slot must be other than zero.

NOTES

NOTES

ALPHABETICAL INDEX

'* axis cannot be synchronized' 42
'* axis feedback error' 45
'* axis following error limit overrun' 43
'* axis hard limit overrun' 43
'* axis locked' 42
'* axis maximum feed exceeded' 42
'* axis range exceeded' 42
'* axis servo error' 43
'* axis soft limit overrun' 43
'* axis work zone 1 overrun' 43
'* axis work zone 2 overrun' 43
'* axis work zone 3 overrun' 44
'* axis work zone 4 overrun' 44
'2nd spindle drive error' 50

A

'A self-intersecting contour has been programmed.' 34
'A subroutine is not allowed for automatic range change' 32
'A tool change has been programmed without M06' 31
'Access to a variable with non-permitted index' 35
'Analog inputs: ANAI(1-8) = +/-5 Volts.' 24
'Analog output not available.' 12
'Analog outputs: ANAO(1-8) = +/-10 Volts.' 24
'Angle coordinate programmed incorrectly' 38
'Arc programmed with radius too small or complete circle' 38
'ASIN/ACOS range exceeded.' 15
'Auxiliary spindle drive error' 50
'Axes board without voltage' 45
'Axes X, Y and Z must exist.' 26
'Axes X, Y or Z slaved or synchronized.' 26
'Axis does not exist.' 9
'Axis drive error on: ' 50

B

'Base zero with positive exponent.' 15
'Beginning of compensation without a straight path' 37
'Block cannot be executed while running another program' 17
'Block incompatible when defining a profile.' 5
'Block not allowed in MDI or during tool inspection' 38

C

'Canned cycle does not exist' 33
'Chamfer programmed incorrectly' 29
'Chamfer value too large' 29
'CHECKSUM ERROR: GENERAL PARAMETERS ' 51
'CHECKSUM ERROR: SPINDLE PARAMETERS ' 51
'CHECKSUM ERROR:2nd SPINDLE PARAMETERS ' 51
'CHECKSUM ERROR:AUX.SPINDLE PARAMETERS ' 51
'CHECKSUM ERROR:AXIS * PARAMETERS ' 51
'CHECKSUM ERROR:CROSS COMP. TABLE 1 ' 52
'CHECKSUM ERROR:CROSS COMP. TABLE 2 ' 52
'CHECKSUM ERROR:CROSS COMP. TABLE 3 ' 52
'CHECKSUM ERROR:LEADSCREW * TABLE ' 52
'CHECKSUM ERROR:M FUNCTION TABLE ' 52
'CHECKSUM ERROR:MAGAZINE TABLE ' 52
'CHECKSUM ERROR:PASSWORD TABLE ' 51
'CHECKSUM ERROR:PLC PARAMETERS ' 51
'CHECKSUM ERROR:SERIAL LINE 1 PARAMETERS ' 51
'CHECKSUM ERROR:SERIAL LINE 2 PARAMETERS ' 51
'CHECKSUM ERROR:TOOL OFFSET TABLE ' 52
'CHECKSUM ERROR:TOOL TABLE " 52
'CHECKSUM ERROR:ZERO OFFSET TABLE ' 51
'Circle with zero radius' 40
'Circular (helical) interpolation not possible.' 24
'Circular path programmed incorrectly' 39
'CNC EPROM memory error' 46
'CNC RAM memory error' 46
'CNC system RAM memory error. Press any key.' 46
'Compensation plane change' 37
'Compensation radius too large' 37
'Complete Table.' 26
'Coupled * axis following error difference too large' 43

D

'Division by zero in PLC.' 48
'Division by zero.' 14
'Do not modify the active tool or the next one.' 19
'Do not program «Q» with parameter M19TYPE=0.' 28
'Do not program a GANTRY axis.' 10
'Do not program a slaved axis.' 10
'Do not program a slaved axis' 41
'Do not program formats greater than 6.5 .' 17
'Do not program labels by parameters.' 3
'Do not switch axes already switched over' 31
'Do not switch axes over or back while G15, G23, G48 or G49 are active' 31
'Do not use high level to change active tool or next one' 31
'Don't program G33 ,G95 or M19 S with no spindle encoder' 23
'Drive error' 49
'Drive overload (201)' 49
'Drive overtemperature (107)' 49

E

'ELSE not associated with IF.' 10
'Empty line.' 1
'End of compensation without a straight path' 37
'Error, undefined class 1' 49
'Errors in taper cycles. ' 55
'Errors in the drilling cycle. ' 60
'Errors in the facing cycle. ' 55
'Errors in the grooving cycles. ' 58
'Errors in the multiple drilling cycle. ' 60
'Errors in the multiple slot milling cycle.' 62
'Errors in the multiple tapping cycle. ' 61
'Errors in the profile cycles of the «C» axis. ' 59
'Errors in the profile cycles. ' 59
'Errors in the rounding cycles. ' 56
'Errors in the tapping cycle. ' 61
'Errors in the threading cycle. ' 57
'Errors in the turning cycle. ' 54
'Expecting "(".' 14
'Expecting ")".' 14
'Expecting ",".' 14
'Expecting "=".' 13
'Expecting a message.' 11
'Expecting a parameter' 12
'External emergency activated' 45

F

'Feedback error (600...606)' 49
'First point programmed wrong when selecting profile' 28
'For G28 or G29, a second spindle is required.' 27
'Format +/- 5.5.' 22
'Function not possible from PLC.' 25

G

'G2 or G3 not allowed when programming a canned cycle.' 5
'G51 [A] E' 18
'G60-G61: X Z I B Q A J [D K H C] S.' 8
'G62-G63: X Z L I Q A J [D] F S.' 8
'G66,G68,G69 not allowed when machining with C axis.' 25
'G66: X Z I C [A L M H] S E.' 7
'G68-G69: X Z C [D L M F H] S E .' 7
'G8 defined incorrectly' 30
'G81-G82: X Z Q R C [D L M F H].' 7
'G83: X Z I B [D K H C].' 7
'G84-85: X Z Q R C [D L M F H] I K.' 6
'G86-87: X Z Q R I B [D L] C [J A].' 6
'G88-G98: X Z Q R [C D K].' 6
'G96 only possible with analog spindle.' 24

H

'Heat-sink overtemperature (106)' 49
'Helical path programmed incorrectly' 39
'High level blocks not allowed when defining a profile.' 5
'HIRTH axis: program only integer values.' 10

I

'I/O 1 board without voltage' 45
'I/O 2 board without voltage' 45
'I/O 3 board without voltage' 45
'Improper data format' 2
'Improper data order.' 1
'Improper data' 1
'Inch programming limit exceeded.' 23
'Incompatible G functions.' 2
'Incompatible tool position and tool code in profile cycle' 34
'Incomplete Coordinates.' 8
'Incomplete operation.' 13
'Incorrect * axis feedrate parameter' 42
'Incorrect * axis leadscrew table. Press any key' 52
'Incorrect access to PLC variables' 34
'Incorrect axis.' 19
'Incorrect cross compensation table 1' 52
'Incorrect cross compensation table 2' 52
'Incorrect cross compensation table 3' 52
'Incorrect cross compensation table parameters' 53
'Incorrect expression.' 13
'Incorrect message.' 22
'Incorrect number of bits.' 22
'Incorrect operation.' 13
'Incorrect order of axes.' 9
'Incorrect parametric programming.' 22
'Incorrect range change' 32
'Incorrect variable value' 34
'Insufficient accelerations for the programmed threadcutting feedrate' 40
'Insufficient memory.' 22
'Invalid cutter geometry angle' 34
'Invalid cutter width' 34
'Invalid G function after first point of profile' 27
'Invalid G function when selecting a profile' 27
'Invalid parameter value in canned cycle' 33
'Invalid programming after first point of profile' 28
'Invalid tool for programmed profile.' 33

J

'Jump to an undefined label' 36

L

'Label cannot be searched' 36
'Label not defined' 36
'Leadscrew: Position-Error.' 19
'Local parameters not accessible' 35
'Local parameters not accessible' 35
'Local parameters not allowed.' 17
'Location code not allowed in canned cycle' 34
'Logarithm of zero or negative number.' 14

M

'M function: M4 S4 bits(8).' 18
'Magazine is not RANDOM.' 20
'Magazine: P(1-255) = T(1-9999).' 19
'Maximum probe travel overrun' 47
'Maximum temperature exceeded' 45
'Modal subroutines cannot be programmed.' 24
'Motor overtemperature (108)' 49

N

'Negative base with decimal exponent.' 15
'Negative radius in polar coordinates' 38
'Nesting exceeded.' 35
'Next tool only possible in machining centers.' 20
'No compensation is permitted.' 27
'No more G functions allowed in the block' 3
'No more information allowed in the block.' 2
'No more M functions allowed in the block' 3
'No negative radius allowed with absolute coordinates' 23
'Nonexistent G function' 3
'Nonparametric assignment after first point of profile' 27
'Not enough information about the path' 29
'Not enough room for the automatic range change M code' 32
'Number of repetitions not possible.' 3
'Numerical format exceeded.' 22

O

'Offset D0 does not exist.' 19
'Offset: D3 X Z R F I K..' 18
'Only one HIRTH axis per block is allowed.' 23
'OPEN is missing.' 11
'Option not available.' 25
'Overcurrent (212)' 49

P

'Parameter does not exist.' 12
'Password: use uppercase/lowercase letters or digits.' 23
'Pitch programmed incorrectly.' 10
'Plane change during rounding or chamfering' 29
'Plane change during tool inspection' 38
'PLC EPROM memory error' 46
'PLC Error -> ' 48
'(PLC_ERR without description)' 48
'PLC not ready.' 46
'PLC RAM error. Press any key.' 46
'PLC RAM memory error' 46
'Point incompatible with active plane.' 9
'Point within the forbidden zone 1' 40
'Point within the forbidden zone 2' 40
'Point within the forbidden zone 3' 41
'Point within the forbidden zone 4' 41
'Polar coordinates not allowed.' 9
'Position-only rotary axis: Absolute values 0 - 359.9999' 23
'Power bus error (213...215)' 49
'Power bus overvoltage (304/306)' 49
'Power bus undervoltage (307)' 49
'Preset of rotary axes: Values between 0-359.9999. ' 26
'Probe signal has not been received' 39
'Probing canned cycle not defined' 35
'Program columns 0 thru 79.' 16
'Program A (append) or D (delete).' 25
'Program A from 0 to 255' 27
'Program already exists.' 12
'Program another softkey.' 15
'Program another window.' 16

'Program axes.' 9
'Program cannot be opened.' 36
'Program channel 0(CNC),1(PLC) or 2(DNC).' 13
'Program column number.' 15
'Program DNC1/2, HD or CARD A (optional).' 24
'Program does not exist.' 12
'Program error number 0 thru 9999.' 13
'Program F, S, T, D before the M functions.' 3
'Program G15 before C axis.' 26
'Program G36-G39 with R+5.5.' 4
'Program INPUT.' 16
'Program inputs 0 thru 25.' 16
'Program inside R' 28
'Program label N(0-9999).' 11
'Program maximum Z' 28
'Program nesting not allowed.' 27
'Program numerical format.' 17
'Program outside R' 28
'Program P3 = value.' 19
'Program pages 0 thru 255.' 16
'Program pitch.' 9
'Program Q between +/-359.9999.' 28
'Program row number.' 15
'Program rows 0 thru 20.' 16
'Program softkeys 1 thru 7.' 15
'Program subroutine number 1 thru 9999.' 11
'Program windows 0 thru 25.' 16
'Program: G16 axis-axis.' 4
'Program: G22 K(1/2/3/4) S(0/1/2).' 4
'Program: G52 axis +/-5.5.' 26
'Program: G72 S5.5 or axes.' 4
'Program: G77 axes (2 thru 6).' 5
'Program: G93 I J.' 5
'Program: work zone K1, K2, K3 or K4.' 4

R

'Radius comp. not possible when positioning rotary axis' 37
'Range exceeded' 39
'Read-only variable.' 12
'Repeated information' 2
'Repeated subroutine.' 11
'Repositioning not allowed.' 25
'RET not associated to a subroutine' 35
'Rotary axis: Absolute values (G90) within +/-359.9999.' 23
'Rounding in last block' 29
'Rounding radius too large ' 29

S

'S has been programmed without an active range' 32
'S has not been programmed in G96' 33
'S not programmed in G95 or threadcutting' 33
'S programmed too large' 32
'SERCOS chip RAM Error. Press a key.' 47
'SERCOS chip version Error. Press a key.' 47
'SERCOS error when homing' 50
'SERCOS ring error 1' 50
'SERCOS ring error' 49
'Sercos variable accessing error' 50
'Spindle drive error' 50
'Spindle feedback error' 45
'Spindle following error limit overrun' 42
'Spindle locked' 41
'Spindle speed range not defined for M19' 32
'Spindle travel limit overrun' 41
'Square root of a negative number.' 14
'Step in a straight path' 37
'Step in circular path' 37
'Subroutine not available in program' 36
'Subroutine not defined' 35

T

'Table limits exceeded.' 18
'Tangential exit programmed incorrectly' 29
'Text too long.' 22
'The axis cannot be programmed after first point of profile' 28
'The canned cycle is missing a tool offset' 31
'The main program cannot have a subroutine.' 11
'The position of a special tool is set.' 20
'The program cannot be executed.' 36
'The program is not accessible' 24
'The radius has not been programmed for G15.' 30
'The Spindle cannot be referenced (homed)' 39
'The tool is not in the tool magazine' 30
'The tracing module has no voltage' 47
'The window must be previously defined.' 24
'There is no C axis.' 25
'There is no empty pocket in the tool magazine' 31
'There is no information on previous path' 30
'There is no live tool.' 25
'There is no subroutine associated with G74' 38
'There is no tool of the same family to replace it' 31
'This command can only be executed in the user channel.' 17
'This G or M function must be alone.' 3
'Tool not defined in tool table' 30
'Tool not defined.' 19
'Tool offset does not exist' 25
'Tool T0 does not exist.' 19
'Tool: T4 D3 F3 N5 R5(.2).' 18

U

'User channel: Do not program geometric aides, comp. or cycles' .. 17
'USER RAM memory error at the CNC. Press any key.' 46

V

'Values 0 thru 100.' 21
'Values 0 thru 2.' 20
'Values 0 thru 255.' 21
'Values 0 thru 3.' 21
'Values 0 thru 32767.' 21
'Values 0 thru 4.' 21
'Values 0 thru 6.' 26
'Values 0 thru 65535.' 21
'Values 0 thru 9.' 21
'Values 0 thru 9999.' 21
'Voltage control error (100...105)' 49

W

'WATCHDOG in Main Module (PRG).' 48
'WATCHDOG in Periodic Module (PE).' 48
'WBUF can only be executed in user channel while editing' 18
'Work zone limit range exceeded' 40
'Write +/-.' 20
'Write 0/1.' 20
'Write ON/OFF.' 20
'Write YES/NO.' 20
'Wrong graphic limits' 28
'Wrong password.' 23
'Wrong plane in tangential path' 30
'Wrong sercosid parameters for axes and spindle' 53
'Wrong tool position prior to canned cycle' 34
'Wrong work zone boundaries' 41

Z

'Zero offset range exceeded' 40
'Zero offset: G54-59 axes (1-5).' 18